Fluid Mechanics and Hydraulic Machinery (DI03000171)

GTU Course Presentation

Table of Contents I

Table of Contents II

Table of Contents III

Table of Contents IV

Table of Contents V

Concept of fluid and its type I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 1: Introduction to Fluid Properties

- Instructor: Mechanical Engineering Professor
- Topics: Concept of Fluid, Shear Stress Response, Types of Fluids, and Density

Concept of Fluid: Definition I

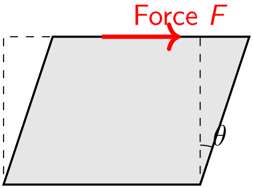
A fluid is a substance that deforms continuously under the application of a shear (tangential) stress, no matter how small that shear stress may be. Unlike solids, which undergo a finite deformation proportional to the applied stress, a fluid reacts by flowing and continues to deform as long as the stress is maintained. This continuous deformation is termed flow, and the rate of deformation (strain rate) is proportional to the applied shear stress in Newtonian fluids.

- Definition: Continuous deformation under tangential shear stress.
- Contrast with Solids: Solids resist shear with static deformation; fluids cannot sustain static shear.
- Flow Behavior: The rate of deformation, rather than the deformation itself, becomes the key parameter.

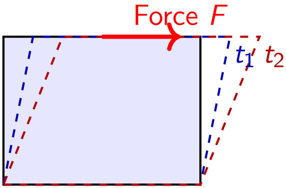
Concept of Fluid: Definition II

- Molecular Spacing: Intermolecular forces are weaker than in solids, allowing molecules to move relative to one another.

Solid vs. Fluid Behavior under Shear I



Solid (Finite θ)



Fluid (Continuous $\theta(t)$)

- Solid: Applied shear force causes a static angular displacement (shear strain, theta).
- Fluid: Applied shear force causes a continuous rate of shear strain (d(theta)/dt).
- No Minimum Stress: Even an infinitesimally small shear force will cause a fluid to flow.
- Boundary Condition: The fluid at the boundary surfaces adheres to them (no-slip condition).

Classification of Fluids I

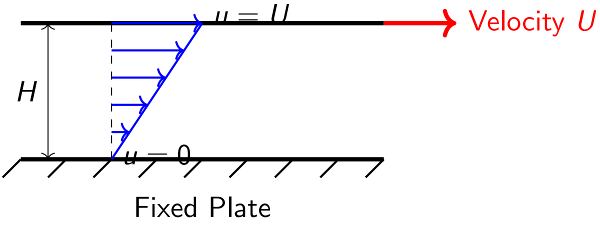
Fluids are broadly categorized based on their response to applied shear stress and their relationship between shear stress and rate of shear deformation. The most fundamental classification divides them into Newtonian and Non-Newtonian fluids. An ideal fluid, which is a theoretical concept, represents a fluid with zero viscosity and absolute incompressibility, whereas real fluids possess viscosity, surface tension, and compressibility.

- Ideal Fluid: Theoretical fluid with zero viscosity, surface tension, and compressibility. It does not offer any resistance to shear.
- Real Fluid: Possesses viscosity, surface tension, and compressibility, meaning it offers resistance when subjected to shear.
- Newtonian Fluid: Obeys Newton's Law of Viscosity (shear stress is directly proportional to velocity gradient).

Classification of Fluids II

- Non-Newtonian Fluid: Does not obey Newton's Law of Viscosity (viscosity varies with shear rate or time).

Newton's Law of Viscosity I



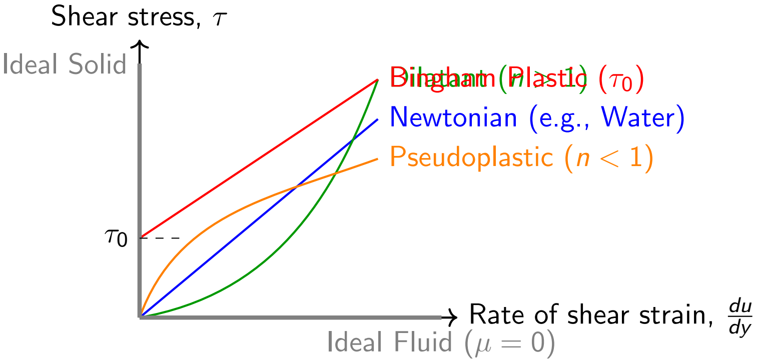
- Newton's Law: $\tau = \mu \frac{du}{dy}$, where τ is the shear stress and μ is the dynamic viscosity.
- Linear Velocity Profile: Under laminar flow between parallel plates, the velocity profile is linear ($u/U = y/H$).
- Velocity Gradient (du/dy): Represents the rate of shear strain or deformation.
- Viscosity (μ): The constant of proportionality representing internal friction or resistance to flow.

Types of Non-Newtonian Fluids I

Non-Newtonian fluids are categorized into time-independent and time-dependent fluids. Time-independent fluids have a shear stress that depends only on the current rate of shear strain, governed by the power-law model $\tau = K(du/dy)^n$. These include dilatant (shear-thickening, $n > 1$) such as cornstarch suspension, pseudoplastic (shear-thinning, $n < 1$) such as blood or paint, and Bingham plastics, which require a yield stress (e.g., toothpaste).

- Dilatant ($n > 1$): Viscosity increases with shear rate (e.g., starch in water, quicksand).
- Pseudoplastic ($n < 1$): Viscosity decreases with shear rate (e.g., polymer solutions, blood).
- Bingham Plastic: Behaves as a solid until a threshold yield stress (τ_0) is exceeded (e.g., sewage sludge, clay).
- Time-Dependent: Viscosity changes over time under constant shear (Thixotropic like paint, Rheopectic like gypsum paste).

Rheological Diagram I



- Visual representation of the relationship between shear stress (τ) and velocity gradient (du/dy).
- Bingham Plastic has an intercept on the y-axis representing the yield stress (τ_0).

Rheological Diagram II

- Slope of the curve at any point represents the apparent viscosity of the fluid.
- Ideal Solid stays on the vertical axis (infinite shear resistance), while Ideal Fluid stays on the horizontal axis (zero shear resistance).

Fluid Properties: Density I

Density (also known as mass density or specific mass) is defined as the mass of a fluid per unit volume. It is denoted by the Greek letter rho (ρ). Mathematically, $\rho = m/V$. For an incompressible fluid, density remains constant with changes in pressure and temperature, while for compressible fluids (such as gases), density varies significantly. In liquid mechanics, water is often taken as a reference fluid with a density of 1000 kg/m^3 at $4^\circ C$ and 1 atm.

- Formula: $\rho = \frac{m}{V}$ with SI units of kg/m^3 and dimensions $[ML^{-3}]$.
- Reference Fluid: Pure water at standard conditions has $\rho \approx 1000$ kg/m^3.
- Factors Influencing Density: Density decreases as temperature increases (expansion), and increases as pressure increases.
- Liquids vs Gases: Liquids are highly incompressible (density changes are negligible), whereas gases are highly compressible.

Specific Weight and Specific Gravity I

Specific weight (or unit weight), denoted by gamma (γ) or w, is the weight of a fluid per unit volume: $\gamma = W/V = \rho g$. Specific gravity (S or SG), also called relative density, is the dimensionless ratio of the density of a fluid to the density of a standard reference fluid (water for liquids, air for gases). This dimensionless property is crucial in hydraulic machinery and hydrostatic calculations.

- Specific Weight (γ): Weight per unit volume, $\gamma = \rho g$, units of $\mathrm{N/m^3}$.
- Specific Gravity (S): Ratio of fluid density to reference fluid density: $S = \rho/\rho_{\text{water}}$ (for liquids).
- Water Reference Value: Water specific weight at $4^\circ C$ is $\gamma \approx 9810\ \mathrm{N/m^3}$.
- Dimensionless Nature: Specific gravity has no dimensions or units, making it universal across unit systems.

Summary and Core Concepts I

In this introductory lecture, we established the fundamental distinction between solids and fluids based on their response to shear stress. We explored the classification of fluids into Newtonian and Non-Newtonian categories based on their rheology, and examined primary fluid properties like density, specific weight, and specific gravity. These concepts form the foundation for analyzing fluid statics, kinematics, and dynamics in hydraulic machinery.

- Fluid definition: Substance that deforms continuously under shear stress.
- Newton's Law of Viscosity: Governs Newtonian fluids where shear stress is proportional to the velocity gradient.
- Non-Newtonian fluids: Can be shear-thinning (pseudoplastic), shear-thickening (dilatant), or require yield stress (Bingham plastic).

Summary and Core Concepts II

- Mass Density (ρ): Ratio of mass to volume, fundamental to all fluid mechanics equations.

Specific Weight, Specific Volume, and Specific Gravity I

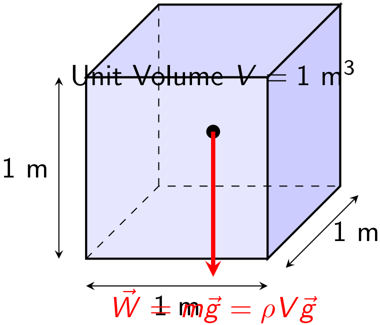
Course: Fluid Mechanics and Hydraulic Machinery Lecture 02: Fundamental Fluid Properties Instructor: Professor of Mechanical Engineering

Introduction to Specific Weight I

Specific weight, also known as unit weight, represents the weight of a fluid per unit volume. Denoted by the Greek letter gamma (γ) or sometimes 'w', it is a fundamental property in fluid mechanics that directly links the mass-based property (density) to the gravitational force acting on the fluid. Unlike density, specific weight is not absolute; it depends on the local gravitational acceleration (g), meaning the specific weight of a fluid will vary slightly depending on elevation and location on Earth, and significantly on other celestial bodies.

- Defined as weight per unit volume: $\gamma =$ W / V
- Mathematically related to density: $\gamma = \rho$ * g
- SI unit is Newtons per cubic meter (N/m^3), dimension is [M L^{-2} T^{-2}]
- Standard value for water at 4°C and standard gravity is 9810 N/m^3 (9.81 kN/m^3)

Visualizing Density vs. Specific Weight I



Mass $m = \rho$
Weight $W = \gamma$

- A unit volume (1 m^3) is used as the reference standard
- Mass contained within this volume is the density (ρ)
- The downward force exerted by gravity on this mass is the specific weight (γ)
- Equation: γ = W/V = (m*g)/V = ρ * g

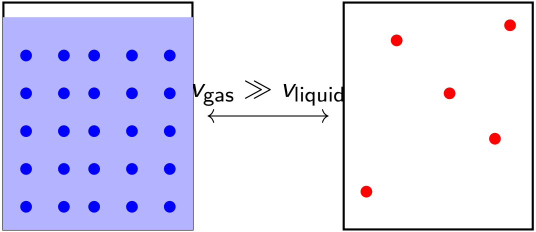
Specific Volume I

Specific volume (v) is defined as the volume occupied by a unit mass of a fluid. It is the reciprocal of mass density (ρ). This property is particularly important in thermodynamics and gas dynamics where compressibility is significant. While liquids have very small and relatively constant specific volumes due to their highly incompressible nature, gases exhibit large and highly variable specific volumes that change dramatically with pressure and temperature according to the equation of state.

- Defined as volume per unit mass: v = V / m
- Reciprocal relation to density: v = 1 / ρ
- SI unit is cubic meters per kilogram (m^3/kg), dimension is [L^3 M^{-1}]
- For water at standard conditions, v is approximately 0.001 m^3/kg

Specific Volume Comparison: Liquid vs. Gas I

For a constant Mass ($m = 1$ kg):



Liquid (Small v, High ρ) Gas (Large v, Low ρ)

- Liquid molecules are closely packed, leading to small volume per unit mass
- Gas molecules are widely spaced, leading to large volume per unit mass
- The diagram contrasts 1 kg of liquid vs 1 kg of gas under standard conditions

Specific Gravity I

Specific gravity (S or SG), also called relative density, is a dimensionless ratio of the density of a fluid to the density of a standard reference fluid. For liquids, the standard reference fluid is pure water at its temperature of maximum density, which occurs at 4°C (where density is exactly 1000 kg/m^3 or specific weight is 9810 N/m^3). For gases, the reference fluid is typically air or hydrogen at standard temperature and pressure (STP). Because it is a ratio, specific gravity is independent of the system of units used.

- Mathematical definition: $S = \rho_{\text{fluid}} / \rho_{\text{ref}} = \gamma_{\text{fluid}} / \gamma_{\text{ref}}$
- For liquids, the reference is water at 4°C: $S = \rho_{\text{liquid}} / 1000$ kg/m^3
- Dimensionless quantity with no units
- Fluids with S ¡ 1 float on water; fluids with S ¿ 1 sink in water

Practical Significance and Engineering Applications I

These specific properties are vital in the analysis and design of hydraulic machinery, pipeline networks, and open channel flows. For instance, specific weight is directly used in hydrostatic pressure calculations ($p = \gamma * h$) to determine force on dams, gates, and submerged surfaces. Specific gravity simplifies calculations by allowing engineers to quickly determine density or specific weight of any fluid from a single ratio, and specific volume is crucial in pump and turbine design to analyze cavitation and fluid compression.

- Hydrostatic pressure: $P = \gamma * h$
- Manometer analysis: relates height difference to pressure using specific gravity
- Buoyancy force calculation: $F_B = \gamma * V_{submerged}$
- Pumping power requirements: $Power = \gamma * Q * H / \eta$

Temperature and Pressure Dependencies I

The specific properties of fluids are not static; they change with thermal and pressure variations. Liquids are generally assumed to be incompressible, meaning their density, specific weight, and specific volume change negligibly with pressure. However, they do expand with temperature, causing specific weight to decrease and specific volume to increase. Gases are highly compressible, and their properties are governed by the Ideal Gas Law: $P = \rho$ R T, which shows specific weight is directly proportional to pressure and inversely proportional to absolute temperature.

- For liquids: T increases -¿ ρ and γ decrease, v increases
- For liquids: P increases -¿ negligible change in ρ, γ, and v
- For gases: T increases (at constant P) -¿ ρ and γ decrease, v increases
- For gases: P increases (at constant T) -¿ ρ and γ increase, v decreases

Worked Example: Specific Weight and Volume Calculations I

Consider a container filled with 8 liters of a liquid that has a mass of 7.2 kg. Let's calculate its density, specific weight, specific volume, and specific gravity. First, convert volume to cubic meters: V = 8 L = 0.008 m^3. Density: ρ = m/V = 7.2 / 0.008 = 900 kg/m^3. Specific Weight: $\gamma = \rho$ * g = 900 * 9.81 = 8829 N/m^3 (8.83 kN/m^3). Specific Volume: v = $1/\rho$ = 1/900 = 0.00111 m^3/kg. Specific Gravity: S = ρ / ρ_{water} = 900 / 1000 = 0.90.

- Given: V = 8 L (0.008 m^3), m = 7.2 kg
- Density (ρ) = 900 kg/m^3
- Specific Weight (γ) = 8829 N/m^3
- Specific Volume (v) = 0.00111 m^3/kg
- Specific Gravity (S) = 0.90 (fluid floats on water)

Summary and Key Formulae I

In this lecture, we have defined three key volume-specific properties of fluids. Understanding these properties forms the bedrock of fluid mechanics, enabling the analysis of fluids at rest (hydrostatics) and in motion (hydrodynamics). In hydraulic machinery, these properties dictate pump selection, conduit sizing, and energy transmission efficiency. Always be mindful of the reference temperature when utilizing specific gravity and the local gravitational acceleration when working with specific weight.

- Specific Weight: $\gamma = \mathsf{W}/\mathsf{V} = \rho * \mathsf{g}$ [N/m^3]
- Specific Volume: $\mathsf{v} = \mathsf{V}/\mathsf{m} = 1/\rho$ [m^3/kg]
- Specific Gravity: $\mathsf{S} = \rho \ / \ \rho_{\text{water at } 4^\circ\text{C}}$ [-]
- Check units and reference standards to avoid dimensional inconsistency

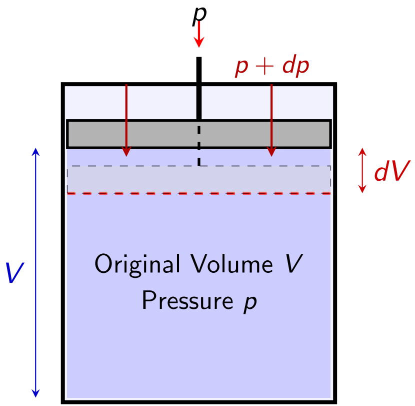
Course: Fluid Mechanics and Hydraulic Machinery Lecture 3: Bulk Modulus, Compressibility, and Dynamic Viscosity

- Introduction to fluid properties governing volume changes and resistance to shear.
- Concepts of elasticity, thermodynamics of compression, and friction in fluid layers.

Compressibility and Bulk Modulus (K) I

Compressibility measures the change in volume (and density) of a fluid when subjected to pressure. The Bulk Modulus of Elasticity (K) is the reciprocal of compressibility, representing a fluid's resistance to compression. It relates the change in hydrostatic pressure to the corresponding volumetric strain.

- Compressibility (β) is mathematically defined as $\beta = -\frac{1}{V}\frac{dV}{dp} = \frac{1}{\rho}\frac{d\rho}{dp}$.
- Bulk Modulus (K) is the reciprocal of compressibility: K $=$ -V$\frac{dp}{dV} = \rho\frac{dp}{d\rho}$.
- The negative sign ensures that the bulk modulus K is positive, as volume decreases (dV ¡ 0) when pressure increases (dp ¿ 0).
- Units: SI unit of Bulk Modulus is Pascal (Pa) or N/m^2, same as pressure. Compressibility is in Pa^{-1}.



- A cylindrical container with a piston containing a fluid under pressure.
- An increase in pressure dp forces the piston down, decreasing the volume by dV.

Diagram: Volumetric Compression of Fluid Element II

- The volumetric strain is represented as -dV/V, and the bulk modulus is K = dp / (-dV/V).

Bulk Modulus of Liquids vs. Gases I

Liquids are generally considered incompressible because their bulk modulus is extremely high. For instance, water has a bulk modulus of approximately 2.2 GPa, meaning a 1

- Water: K $\approx 2.2 \times 10^9$ Pa; Air (at standard conditions): K $\approx 1.01 \times 10^5$ Pa.
- Because K of liquids is so large, they are treated as incompressible ($\rho =$ constant) in most hydraulic applications.
- Exceptions in liquids include high-pressure systems, water hammer analysis, and acoustic wave propagation where compressibility is vital.

Thermodynamic Variations of Bulk Modulus in Gases I

For gases, the relationship between pressure and density depends on the thermodynamic path. Under isothermal conditions (constant temperature), the bulk modulus equals the absolute pressure. Under isentropic conditions (reversible adiabatic), the bulk modulus equals the product of the specific heat ratio (gamma) and the absolute pressure.

- Isothermal process: $\mathsf{p}/\rho = \mathsf{C} \implies \mathsf{K_T} = \mathsf{p}$.
- Isentropic process: $\mathsf{p}/\rho^{\gamma} = \mathsf{C} \implies \mathsf{K_s} = \gamma\, \mathsf{p}$.
- For air ($\gamma = 1.4$), the adiabatic bulk modulus is 40
- Acoustic waves propagate isentropically through gases, making $\mathsf{K_s}$ the relevant parameter for the speed of sound ($\mathsf{c} = \sqrt{K_s/\rho}$).

Speed of Sound in Fluids I

The speed of sound (c) in a fluid is directly related to its bulk modulus and density. It represents the velocity at which pressure disturbances propagate through the medium. Using the Newton-Laplace equation, the speed of sound is the square root of the ratio of the bulk modulus to the fluid density.

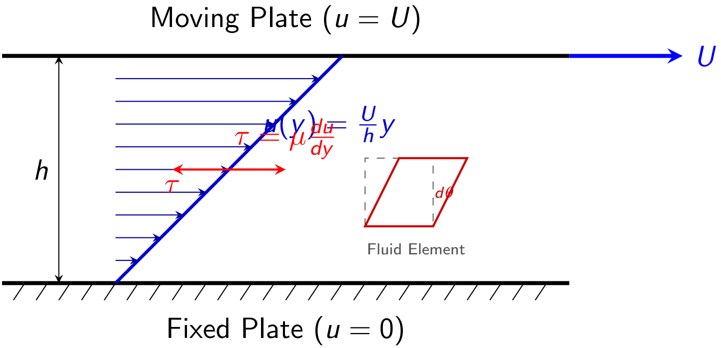
- Acoustic wave speed equation: $c = \sqrt{\frac{K}{\rho}}$.
- In water: $c = \sqrt{2.2 \times 10^9 \text{ Pa}/1000 \text{ kg}/\text{m}^3} \approx 1483 \text{ m/s}$.
- In air (isentropic, 15° C): $c = \sqrt{1.4 \times 101325 \text{ Pa}/1.225 \text{ kg}/\text{m}^3} \approx 340 \text{ m/s}$.
- Higher bulk modulus leads to faster propagation speeds, which is why sound travels faster in liquids and solids than in gases.

Introduction to Dynamic Viscosity (μ) I

Dynamic viscosity (μ) is the fluid property that measures its resistance to gradual deformation by shear or tensile stress. It represents the internal friction of a moving fluid. According to Newton's Law of Viscosity, the shear stress between fluid layers is directly proportional to the rate of shear strain (velocity gradient).

- Newton's Law of Viscosity: $\tau = \mu \, \frac{du}{dy}$.
- τ is shear stress (N/m^2), μ is dynamic viscosity (Pa·s), and du/dy is the velocity gradient (s^{-1}).
- SI Unit: Pascal-second (Pa·s) or Kilogram per meter-second (kg/(m·s)). Common cgs unit is Poise (P), where 1 Pa·s = 10 Poise.
- Viscosity is caused by cohesive forces between molecules (liquids) and molecular momentum transfer (gases).

Diagram: Velocity Profile & Shear Stress in Fluid Flow I



- Velocity profile of a Newtonian fluid undergoing shear flow between parallel plates.
- The velocity gradient du/dy is constant in laminar, fully developed Couette flow.

Diagram: Velocity Profile & Shear Stress in Fluid Flow II

- Viscous shear stress acts along fluid layers, opposing relative motion and generating heat.

Temperature Dependence of Viscosity I

The dynamic viscosity of fluids is strongly dependent on temperature, but liquids and gases exhibit opposite behaviors. For liquids, viscosity decreases with an increase in temperature due to the reduction of cohesive intermolecular forces. For gases, viscosity increases with temperature because higher temperatures enhance molecular collisions and momentum transfer.

- Liquid viscosity variation can be approximated by Andrade's equation: $\mu = \mathrm{A}\ \mathrm{e}^{\mathrm{B}/\mathrm{T}}$.
- Gas viscosity variation is commonly modeled using Sutherland's law: $\mu = \mu_0\ (\frac{T}{T_0})^{3/2}\ \frac{T_0+S}{T+S}$.
- Engineers must carefully account for temperature when designing hydraulic circuits and lubrication systems.
- For example, engine oil viscosity is critical across wide operational temperature ranges (e.g., cold start vs. high-temperature highway driving).

Summary and Hydraulic Engineering Applications I

Bulk modulus, compressibility, and viscosity are fundamental parameters in the design of hydraulic systems, turbines, and aerospace vehicles. Controlling water hammer in pipelines relies on understanding bulk modulus, while lubrication efficiency and pipeline pressure losses are governed by dynamic viscosity.

- Water hammer (pressure surges) occurs when a moving fluid is forced to stop suddenly, converting kinetic energy into elastic strain energy governed by K.
- Hydraulic power transmission requires high bulk modulus fluid to minimize energy loss and ensure fast response times.
- Dynamic viscosity dictates the Reynolds number ($Re = \rho$ v D / μ), determining whether flow is laminar or turbulent.
- Pumping power requirements are directly proportional to the fluid viscosity due to frictional pressure drops.

Lecture 4: Kinematic Viscosity, Surface Tension & Capillarity I

Course: Fluid Mechanics and Hydraulic Machinery (DI03000171). This lecture covers the fundamental physical properties of fluids, focusing on kinematic viscosity, surface tension, and capillarity.

- Instructor: Professor of Mechanical Engineering
- Topics: Kinematic Viscosity, Surface Tension, Capillarity
- Prerequisites: Density, Dynamic Viscosity, Newton's Law of Viscosity

Understanding Kinematic Viscosity I

Kinematic viscosity (ν) represents a fluid's resistance to shear flow under the influence of gravity. It is mathematically defined as the ratio of dynamic viscosity (μ) to the fluid density (ρ): $\nu = \mu/\rho$. While dynamic viscosity measures the internal friction force per unit area, kinematic viscosity measures the rate at which momentum diffuses through the fluid due to molecular motion, acting as momentum diffusivity.

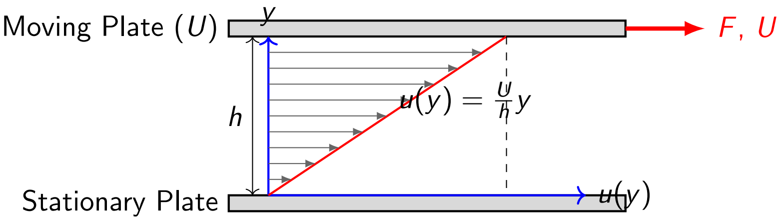
- Defined as the ratio of dynamic viscosity to density: $\nu = \frac{\mu}{\rho}$.
- Physical interpretation: Momentum diffusivity of the fluid.
- SI unit: $\mathrm{m^2/s}$. CGS unit: Stokes (St), where $1\ \mathrm{St} = 10^{-4}\ \mathrm{m^2/s}$ and $1\ \mathrm{cSt} = 10^{-6}\ \mathrm{m^2/s}$.
- Unlike dynamic viscosity, it accounts for fluid density, making it crucial in gravity-driven flows.

Physical Significance & Dimensional Analysis I

The dimensional formula for kinematic viscosity is $[L^2T^{-1}]$, which is identical to thermal diffusivity and mass diffusivity. This similarity underscores its role in the transport of momentum. In fluid dynamics, the Reynolds number ($Re = VD/\nu$) uses kinematic viscosity directly to predict the transition from laminar to turbulent flow. Water, despite being much denser than air, has a lower kinematic viscosity than air at standard conditions, meaning air diffuses momentum faster than water.

- Dimensional formula: $[L^2T^{-1}]$.
- Governs the Reynolds number ($Re = \frac{VD}{\nu}$), representing the ratio of inertial to viscous forces.
- Direct analogy to thermal diffusivity (α) and mass diffusivity (D_{AB}) in transport phenomena.
- At $20^\circ C$, $\nu_{\text{air}} \approx 1.5 \times 10^{-5}$ m^2/s while $\nu_{\text{water}} \approx 1.0 \times 10^{-6}$ m^2/s.

Momentum Diffusion & Velocity Profile I



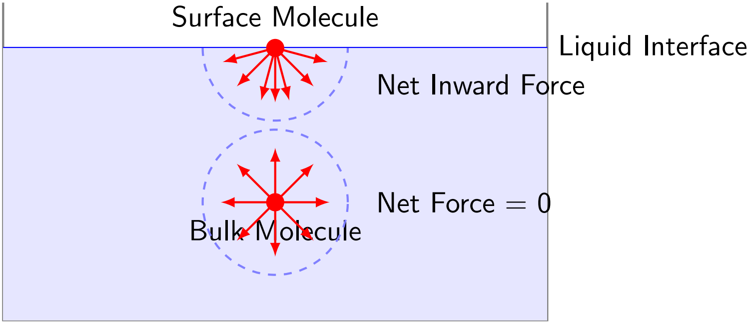
- Visual representation of linear velocity distribution in laminar shear flow.
- Boundary conditions: No-slip condition at both solid boundaries (zero relative velocity).
- Momentum diffusion coefficient (kinematic viscosity ν) governs the rate of development of this velocity profile.

Surface Tension: Molecular Origin I

Surface tension (σ) is a binary property existing at the interface of two immiscible phases. At the molecular level, a molecule within the bulk liquid is attracted equally in all directions by neighboring molecules, resulting in zero net force. However, a molecule at the surface experiences a net inward cohesive force tending to pull it into the liquid. This force pulls the surface molecules together, causing the surface to behave like a stretched elastic membrane.

- Originates from cohesive intermolecular forces between liquid molecules.
- Bulk molecules experience isotropic forces, while surface molecules experience a net inward force.
- Defined as force per unit length: $\sigma = \frac{F}{L}$ (N/m), or energy per unit area: J/m^2.
- Governed by temperature: surface tension decreases as temperature increases due to increased molecular activity.

Molecular Cohesion and Surface Energy I



- Unbalanced force distribution creates high surface energy density.
- The bulk molecule is in static equilibrium due to symmetrical intermolecular attraction.
- The surface molecule experiences an inward pull, minimizing the surface area of the liquid droplet.

Surface Tension: Applications & Mathematical Formulations I

Surface tension creates an internal pressure difference across curved interfaces, described by the Young-Laplace equation. For a spherical liquid droplet, the internal pressure exceeds external pressure by $\Delta P = 2\sigma/R$. For a hollow soap bubble (which has two interfaces, inner and outer), the excess pressure is doubled to $\Delta P = 4\sigma/R$. For a cylindrical liquid jet, the excess pressure is $\Delta P = \sigma/R$. These pressure differences prevent droplets and bubbles from collapsing.

- Liquid Droplet: Excess pressure $\Delta P = \frac{2\sigma}{R}$ where R is the droplet radius.
- Soap Bubble: Excess pressure $\Delta P = \frac{4\sigma}{R}$ due to two active surface interfaces.
- Liquid Jet: Excess pressure $\Delta P = \frac{\sigma}{R}$.
- Practical applications include droplet formation in fuel injection and ink-jet printing.

Capillarity: Cohesion vs Adhesion I

Capillarity is the rise or fall of a liquid in a narrow tube, caused by the relative strength of cohesive forces (molecular attraction within the liquid) and adhesive forces (attraction between liquid molecules and the tube wall). If adhesive forces dominate (e.g., water on clean glass), the liquid wets the wall, creating a concave meniscus and causing capillary rise. If cohesive forces dominate (e.g., mercury on glass), the liquid does not wet the wall, resulting in a convex meniscus and capillary depression.

- Wetting fluid (Contact angle $\theta < 90^\circ$): Capillary rise (e.g., water on glass).
- Non-wetting fluid (Contact angle $\theta > 90^\circ$): Capillary depression (e.g., mercury on glass).
- Contact angle θ represents the angle of contact at the solid-liquid-gas junction.

Capillarity: Cohesion vs Adhesion II

- Important in groundwater transport, soil moisture retention, and wick-based lubrication.

Derivation of Capillary Rise/Fall I

To determine the height (h) of capillary rise/fall, we perform a force balance along the vertical axis. The upward force is the vertical component of the surface tension force acting along the perimeter of the tube: $F_{\text{up}} = \sigma \cdot \pi d \cdot \cos\theta$. The downward force is the weight of the column of liquid supported: $F_{\text{down}} = \rho g \cdot (\pi d^2/4) \cdot h$. Equating these forces yields the capillary height equation: $h = \frac{4\sigma\cos\theta}{\rho g d}$.

- Force balance: Upward surface tension component = Downward weight of the liquid column.
- Capillary height formula: $h = \frac{4\sigma\cos\theta}{\rho g d}$, where d is the tube inner diameter.
- Height is inversely proportional to tube diameter ($h \propto 1/d$); significant only for small diameters ($d < 10$ mm).
- For pure water on clean glass, contact angle $\theta \approx 0^\circ$, hence $\cos\theta \approx 1$.

Engineering Problems & Summary I

This lecture established the foundations of kinematic viscosity, surface tension, and capillarity. We learned how viscosity acts as momentum diffusivity, how surface tension generates pressure jumps across interfaces, and how capillary action governs fluid behavior in microchannels and porous media. In engineering applications, capillary rise must be accounted for in manometer readings to avoid measurement errors, typically requiring tube diameters larger than 10 mm to minimize capillary effects.

- Kinematic Viscosity governs momentum transport and Reynolds number classification.
- Surface tension creates pressure gradients across curved liquid interfaces (droplets, bubbles, jets).
- Capillary rise/fall varies inversely with tube diameter, necessitating correction in precision instruments.

Engineering Problems & Summary II

- Next Lecture: Fluid Statics - Pressure measurement techniques and hydrostatic forces on surfaces.

Vapor Pressure I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 5: Vapor Pressure, Cavitation, and Fluid Pressure Measurement

Molecular Mechanism of Vapor Pressure I

Vapor pressure is the pressure exerted by a vapor in thermodynamic equilibrium with its condensed phases (solid or liquid) at a given temperature in a closed system. At the molecular level, liquid molecules possess a distribution of kinetic energies. Those near the free surface with kinetic energy exceeding the intermolecular attractive forces escape into the space above, a process known as vaporization. Concurrently, vapor molecules collide with the liquid surface and re-enter the liquid phase (condensation). Dynamic equilibrium is reached when the rate of vaporization equals the rate of condensation, resulting in a constant vapor pressure.

- Vapor pressure is a function of temperature only, increasing non-linearly with temperature described by the Clausius-Clapeyron equation.

Molecular Mechanism of Vapor Pressure II

- Boiling occurs when the local vapor pressure of the liquid equals or exceeds the surrounding external atmospheric pressure.
- Volatile liquids (e.g., ether, gasoline) possess high vapor pressures at room temperature due to weak intermolecular forces.

Cavitation in Hydraulic Systems I

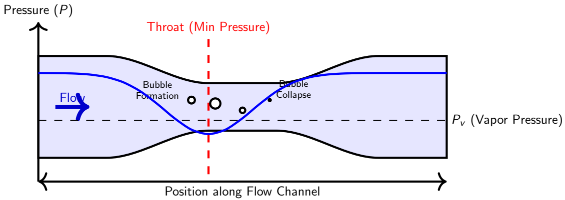
Cavitation is a critical phenomenon in hydraulic machinery where the local pressure drops below the vapor pressure of the liquid at the operating temperature, causing localized boiling and the formation of vapor bubbles (cavities). As these bubbles flow into regions of higher pressure, they collapse or implode violently. The implosion occurs in milliseconds, generating localized micro-jets with velocities of hundreds of meters per second and shockwaves exceeding pressures of several GPa. This leads to severe material erosion (pitting), noise, vibration, and a significant drop in the efficiency of the hydraulic machine.

- Common locations for cavitation include the suction side of pump impellers, turbine runner blades, and low-pressure regions in venturi tubes.

Cavitation in Hydraulic Systems II

- Cavitation damage is characterized by mechanical fatigue and erosion rather than chemical corrosion, though it can accelerate chemical attack.
- The cavitation parameter (Thomas cavitation coefficient) is used to predict the onset of cavitation in hydraulic machinery.

Cavitation Mechanism I



- As fluid velocity increases at the constriction (throat), static pressure drops according to Bernoulli's principle.
- If static pressure drops below the vapor pressure (P_v), vapor bubbles form.
- Downstream, pressure recovers, causing bubbles to collapse violently near solid boundaries.

Fluid Pressure and Pascal's Law I

Fluid pressure is defined as the normal force exerted by a fluid per unit area on any surface in contact with it. Unlike solids, static fluids cannot sustain shear stresses; hence, the pressure force must act perpendicular to any boundary surface. Pascal's Law states that the pressure at any point in a fluid at rest is equal in all directions. Mathematically, for a small tetrahedral fluid element in equilibrium, the pressure forces balancing in the Cartesian axes prove that $Px = Py = Pz$. This principle underpins hydraulic systems, where a small force applied on a small area is transmitted as a large force on a larger area.

- Pressure is a scalar quantity, although the force it generates on a surface is a vector directed normal to the surface.
- In a continuous fluid at rest, pressure remains constant along any horizontal plane.

- Hydrostatic pressure transmission is instantaneous and lossless in ideal, incompressible fluids.

Hydrostatic Pressure Distribution I

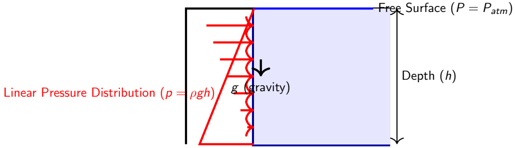
The variation of pressure in a static fluid subject to gravity is governed by the hydrostatic equation, derived from the force balance on a differential fluid element. For an incompressible fluid of constant density (rho), the change in pressure (dp) with respect to height (dz) is given by dp/dz = -rho*g, where g is the acceleration due to gravity and the vertical coordinate z is measured upwards. Integrating this from a free surface at z = h (where pressure is atmospheric, Patm) down to a depth y yields the classic hydrostatic pressure equation: P = Patm + rho*g*y, indicating that pressure increases linearly with depth.

- The term rho*g*y is known as the gauge pressure, representing the pressure relative to the atmospheric datum.
- The pressure distribution is independent of the shape or total volume of the container, a phenomenon known as the Hydrostatic Paradox.

Hydrostatic Pressure Distribution II

- For compressible fluids like gases, density varies with pressure, requiring an equation of state (e.g., $P = \text{rho*R*T}$) to integrate the hydrostatic equation.

Hydrostatic Pressure Prism I



- Pressure increases linearly with depth, resulting in a triangular pressure prism acting on the vertical wall.
- The magnitude of the resultant hydrostatic force is equal to the area of the pressure diagram multiplied by the width of the wall.
- The center of pressure (point of action of the resultant force) is located at a depth of 2/3 from the free surface for a rectangular vertical wall.

Measurement of Fluid Pressure I

Pressure measurement devices are classified based on their operating principles: manometers, which use liquid columns to balance pressure forces, and mechanical gauges, which utilize elastic deformation. U-tube manometers measure pressure differences by observing the deflection of a manometric fluid of high density (such as mercury). Differential manometers measure the difference in pressure between two points in a piping system. Mechanical gauges, such as the Bourdon tube, diaphragm gauges, and bellows, translate the elastic deformation of a metal element under pressure into mechanical pointer movement or digital signals.

- Manometers are highly accurate for low to moderate pressure ranges and do not require calibration, but they have slow dynamic response.

Measurement of Fluid Pressure II

- Bourdon tube gauges are robust and suitable for high-pressure measurements but require periodic calibration against dead-weight testers.
- Piezoelectric and piezoresistive pressure transducers are used for high-frequency dynamic pressure measurements.

Numerical Problem 1: Cavitation Check I

A siphon is used to draw water (density rho = 1000 kg/m^3, vapor pressure Pv = 2.34 kPa absolute) from a reservoir. The highest point of the siphon is 4.5 m above the reservoir surface. If the atmospheric pressure is 101.3 kPa, determine if cavitation will occur at the summit when the water velocity at that point is 5 m/s. Neglect frictional losses.
Solution: Apply Bernoulli's Equation between the reservoir free surface (1) and the summit (2): $P1/(rho*g) + v1^2/(2g) + z1 = P2/(rho*g) + v2^2/(2g) + z2$ Here, P1 = 101.3 kPa absolute, v1 = 0, z1 = 0, z2 = 4.5 m, v2 = 5 m/s, and g = 9.81 m/s^2. $P2 = P1 - rho*g*z2 - 0.5*rho*v2^2$ P2 = 101,300 - (1000 * 9.81 * 4.5) - (0.5 * 1000 * 5^2) P2 = 101,300 - 44,145 - 12,500 = 44,655 Pa = 44.66 kPa absolute.

- Since P2 = 44.66 kPa absolute ¿ Pv = 2.34 kPa absolute, the local pressure is above the vapor pressure.

- Therefore, cavitation will not occur at the summit under these operating conditions.
- If the elevation of the summit is increased or the velocity is higher, the local pressure would drop, increasing the risk of cavitation.

Numerical Problem 2: U-Tube Manometer I

A U-tube manometer containing mercury (specific gravity Sm = 13.6) is connected to a pipe carrying water (specific gravity Sw = 1.0). The center of the pipe is at the same level as the mercury-water interface in the left limb. The mercury level in the right limb is 250 mm above that in the left limb, and the right limb is open to the atmosphere (Patm = 101.3 kPa). Calculate the gauge pressure and absolute pressure of the water in the pipe. Solution: Let the pressure in the pipe be Pp. Equate pressures in both limbs at the level of the mercury-water interface in the left limb (datum line): Left Limb Pressure: PL = Pp Right Limb Pressure: PR = Patm + rho_m*g*h where h = 0.25 m, rho_m = 13.6 * 1000 = 13,600 kg/m^3, and g = 9.81 m/s^2. Pp (absolute) = 101,300 + (13600 * 9.81 * 0.25) = 101,300 + 33,354 = 134,654 Pa = 134.65 kPa absolute. Pp (gauge) = Pp (absolute) - Patm = 33.35 kPa gauge.

Numerical Problem 2: U-Tube Manometer II

- Equating pressures at the lowest common fluid interface (datum line) simplifies manometric calculations.
- The gauge pressure is directly proportional to the deflection height and the difference in density between the manometric and process fluids.
- A higher deflection indicates higher process fluid pressure relative to the atmosphere.

Lecture 6: Pressure Head & Pascal's Law I

Course: Fluid Mechanics and Hydraulic Machinery. This lecture covers the fundamental concepts of pressure head, the mathematical derivation and physical implications of Pascal's Law, and the relationship between absolute, gauge, and vacuum pressures.

- Instructor: Professor of Mechanical Engineering
- Topics: Pressure Head, Pascal's Law, Pressure Scales
- Prerequisites: Fluid Properties, Hydrostatic Pressure Definition

Concept of Pressure in Fluids I

Fluid pressure is defined as the compressive force per unit area acting normal to any surface within or containing the fluid. Unlike solids, static fluids cannot support shear stress, meaning that pressure forces always act perpendicular to boundary surfaces. At any point within a static fluid, the pressure must be isotropic, a property known as hydrostatic pressure.

- Defined mathematically as $P = \lim_{\Delta A \to 0} \frac{\Delta F}{\Delta A}$, where ΔF is the normal force.
- SI unit is the Pascal (Pa), equivalent to 1 N/m^2, often expressed in kilopascals (kPa) or megapascals (MPa).
- Pressure is a scalar quantity; it has magnitude but no directional preference until it acts on a surface.

Derivation of the Hydrostatic Equation I

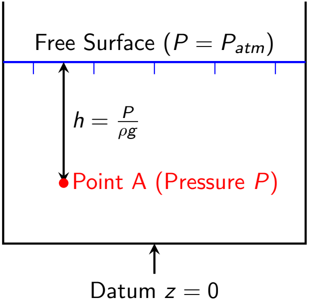
By considering a cylindrical fluid element of cross-sectional area dA and height dz at static equilibrium, we balance the forces in the vertical direction. The downward forces are gravity (weight of the element) and the pressure force at the top, while the upward force is the pressure force at the bottom. This yields the fundamental equation of fluid statics.

- Vertical equilibrium:
  $\sum F_z = 0 \implies P \cdot dA - (P + dP) \cdot dA - \rho \cdot g \cdot dA \cdot dz = 0.$
- Simplifying the force balance gives the differential form:
  $\frac{dP}{dz} = -\rho g = -\gamma$, where γ is the specific weight.
- Integrating for an incompressible fluid ($\rho = \text{const}$) gives $P_2 - P_1 = -\rho g(z_2 - z_1)$, indicating pressure increases linearly with depth.

The Pressure Head Concept I

Pressure head represents the height of a static column of a specific fluid required to produce a given pressure. It is a highly convenient way to express pressure in hydraulic engineering because it relates pressure directly to potential energy per unit weight of the fluid. The relationship is derived directly from the integrated hydrostatic equation, assuming a free surface at atmospheric pressure.

- Formula: $h = \frac{P}{\rho g} = \frac{P}{\gamma}$, where h is the pressure head in meters of the fluid.
- Enables intuitive visualization of energy levels in hydraulic systems, contributing to the hydraulic grade line (HGL).
- Conversion depends on fluid density: a pressure of 101.3 kPa corresponds to 10.3 m of water but only 760 mm of mercury.



- The diagram represents a container with a static fluid having a free surface at atmospheric pressure.
- At depth h below the free surface, the gauge pressure is given by $P = \rho g h$.
- The height h is the pressure head, showing how vertical position determines pressure in static liquids.

Pascal's Law Formulation I

Pascal's Law states that pressure applied to a confined fluid is transmitted undiminished to every portion of the fluid and the walls of the containing vessel. It forms the foundational principle for all hydraulic machinery, including hydraulic jacks, brakes, and presses, by allowing force multiplication through area ratios.

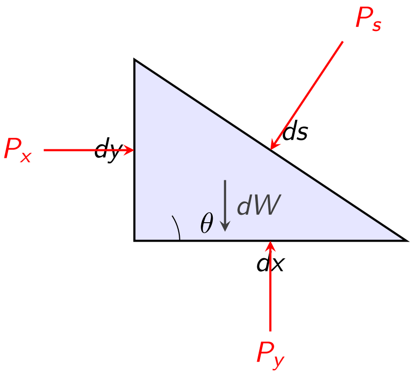
- Mathematically, in an enclosed static fluid, any change in pressure ΔP is transmitted equally throughout.
- For two connected pistons of areas A_1 and A_2, the force transmission obeys $F_1/A_1 = F_2/A_2$.
- Mechanical advantage ($M.A. = F_2/F_1 = A_2/A_1$) allows a small force to lift a substantial load.

Mathematical Derivation of Pascal's Law I

To prove that pressure at a point in a static fluid is equal in all directions, we analyze a tiny, wedge-shaped fluid element of unit width perpendicular to the page. The forces acting on the element are the pressure forces on its three surfaces and the gravitational force (weight). Since the fluid is static, the sum of forces in both horizontal (x) and vertical (y) directions must be zero.

- Horizontal force balance: $P_x \cdot dy \cdot 1 - P_s \cdot ds \cdot 1 \cdot \sin\theta = 0$. Since $ds \sin\theta = dy$, we get $P_x = P_s$.
- Vertical force balance:
  $P_y \cdot dx \cdot 1 - P_s \cdot ds \cdot 1 \cdot \cos\theta - \frac{1}{2}\rho g \cdot dx \cdot dy \cdot 1 = 0$.
- Since $ds \cos\theta = dx$ and the third term (weight) is of higher order ($dx \cdot dy$), it vanishes as the element shrinks to a point, yielding $P_y = P_s$.
- Conclusion: $P_x = P_y = P_s$, proving pressure is isotropic at a point in a static fluid.

Wedge Element Force Balance I



- The triangular prism element has dimensions dx, dy, and hypotenuse ds, with a unit thickness perpendicular to the plane.

Wedge Element Force Balance II

- Forces acting on the faces are due to normal pressures P_x, P_y, and P_s, acting perpendicular to their respective surfaces.
- In the limit as the wedge size approaches zero, the weight element $dW = \frac{1}{2}\rho g dx dy$ becomes negligible, proving $P_x = P_y = P_s$.

Absolute vs. Gauge vs. Vacuum Pressure I

Pressure measurements are classified based on their reference datum. Absolute pressure is measured relative to a perfect thermodynamic vacuum (zero pressure reference). Gauge pressure is measured relative to the local atmospheric pressure, representing pressures above atmospheric level. Vacuum pressure represents pressure levels below local atmospheric pressure, measured downwards from the atmospheric datum.

- Gauge Pressure relationship: $P_{\text{gauge}} = P_{\text{abs}} - P_{\text{atm}}$ (positive when absolute pressure is greater than atmospheric).
- Vacuum Pressure relationship: $P_{\text{vacuum}} = P_{\text{atm}} - P_{\text{abs}}$ (positive when absolute pressure is less than atmospheric).
- In engineering calculations, unless specified as absolute (psia, Pa abs), pressure is typically assumed to be gauge pressure.

Summary and Industrial Applications I

Understanding pressure head, isotropic transmission via Pascal's law, and reference scales is vital for analyzing hydraulic systems. Hydraulic jacks leverage Pascal's Law to amplify input forces, while manometers and Bourdon gauges utilize the hydrostatic principle of pressure head to measure system pressure levels accurately relative to local atmospheric conditions.

- Pressure head ($h = P/\gamma$) simplifies energy equations (Bernoulli's Equation) in civil and mechanical piping design.
- Pascal's Law is the operating principle behind hydraulic presses, heavy machinery actuators, and automotive braking systems.
- Properly distinguishing between gauge and absolute pressure prevents severe errors in thermodynamic calculations and vacuum pump selection.

Course: Fluid Mechanics and Hydraulic Machinery Lecture 7: Atmospheric Pressure, Absolute Pressure, and Numerical Problems

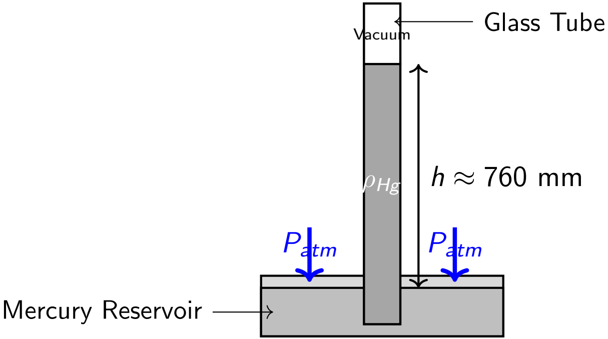
- Core Concepts of Atmospheric Pressure and its physical origins
- Understanding Absolute, Gauge, and Vacuum Reference Planes
- Analytical solutions to pressure measurement problems

Concept of Atmospheric Pressure I

Atmospheric pressure (P_{atm}) is the force per unit area exerted on a surface by the weight of air above that surface in the Earth's atmosphere. At sea level, standard atmospheric pressure is defined as 101.325 kPa, 1.01325 bar, 760 mmHg, or 10.3 m of water. The density of air decreases exponentially with altitude, which causes atmospheric pressure to also decrease non-linearly with altitude. The standard atmosphere (ISA) model provides a reference for aerodynamic and thermodynamic calculations.

- Equivalent to 101,325 N/m^2 or 14.696 psi at mean sea level.
- Measured using a mercury barometer, first designed by Evangelista Torricelli in 1643.
- Highly dependent on local temperature, altitude, and weather patterns.
- Critical parameter in fluid mechanics as it establishes the local reference datum for pressure measurements.

Measurement of Atmospheric Pressure: Torricelli's Barometer I



- A glass tube closed at one end is filled with mercury and inverted into a mercury reservoir.
- The mercury column drops until the hydrostatic pressure of the column equals the atmospheric pressure.

Measurement of Atmospheric Pressure: Torricelli's Barometer II

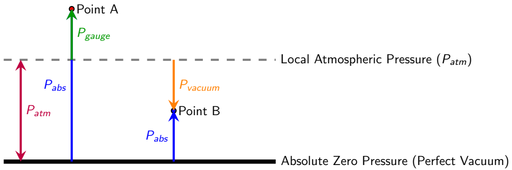
- The space above the mercury column is a near-vacuum containing only mercury vapor (negligible vapor pressure, 0.173 Pa at 20°C).
- Hydrostatic equilibrium equation: $P_{atm} = \rho * g * h$, where ρ is density of mercury (13,560 kg/m^3).

Pressure Reference Planes: Absolute and Gauge I

Pressure measurements require a reference datum. Absolute pressure (P_{abs}) is measured relative to a perfect vacuum (absolute zero pressure). Gauge pressure (P_g) is measured relative to the local atmospheric pressure. If the pressure is below local atmospheric pressure, it is called vacuum pressure (P_{vac}). The mathematical relationships are straightforward but crucial for avoiding errors in thermodynamic and fluid dynamics calculations.

- Absolute Pressure formula: $P_{abs} = P_{atm} + P_g$ (for pressures above atmospheric).
- Vacuum Pressure formula: $P_{vac} = P_{atm} - P_{abs}$ (for pressures below atmospheric).
- Standard pressure gauges read zero when open to the atmosphere, meaning they measure gauge pressure.
- Absolute pressure must always be used in thermodynamic equations of state (e.g., ideal gas law $PV = mRT$).

Relations Between Pressure References I



- The baseline represents Absolute Zero Pressure (Perfect Vacuum), which is a fixed reference.
- The local atmospheric pressure level varies with time and altitude, serving as a floating reference.
- Gauge pressure extends upward from the atmospheric pressure line.
- Vacuum pressure extends downward from the atmospheric pressure line to indicate suction or negative gauge pressure.

Mathematical Definition and Sign Conventions I

In engineering calculations, the sign of gauge pressure dictates whether a system is pressurized or evacuated. A positive gauge pressure (P_g ¿ 0) implies the system's absolute pressure exceeds the ambient atmospheric pressure. A negative gauge pressure (P_g ¡ 0), often expressed as a positive vacuum pressure, implies the system's absolute pressure is less than ambient. Correctly identifying these references prevents catastrophic sign errors in control volume analysis and turbomachinery design.

- If P_{abs} ¿ P_{atm}, Gauge Pressure = P_{abs} - P_{atm}.
- If P_{abs} ¡ P_{atm}, Vacuum Pressure = P_{atm} - P_{abs} (often written as negative gauge).
- Absolute pressure is always positive (P_{abs} ¿= 0) in classical fluid mechanics.
- Atmospheric pressure varies with altitude: $P_{atm}(z) = P_0 * (1 - L*z/T_0)^{(g*M/(R*L))}$.

Numerical Problem 1: Air Receiver Tank I

A pressure gauge connected to an air receiver tank reads 5.8 bar. If the local atmospheric pressure is 750 mmHg, determine the absolute pressure in the tank. Take the density of mercury as 13,600 kg/m^3 and $g = 9.81\ m/s^2$.

- Step 1: Calculate local atmospheric pressure from barometer reading. $P_{atm} = \rho_{Hg} * g * h_{Hg} = 13600 * 9.81 * 0.750 = 100{,}062\ Pa = 1.0006\ bar$.
- Step 2: Note the given gauge pressure: $P_{gauge} = 5.8\ bar = 580{,}000\ Pa$.
- Step 3: Apply absolute pressure formula: $P_{abs} = P_{atm} + P_{gauge}$.
- Step 4: Compute final value: $P_{abs} = 1.0006\ bar + 5.8\ bar = 6.8006\ bar$ (or 680.06 kPa).

Numerical Problem 2: Steam Condenser Vacuum I

A vacuum gauge attached to a steam condenser reads 620 mmHg. The local barometer reads 765 mmHg. Calculate the absolute pressure inside the condenser in kPa. Take the density of mercury as 13,560 kg/m^3 and g = 9.81 m/s^2.

- Step 1: Express both vacuum and atmospheric pressure in terms of mercury column height: h_{vac} = 620 mmHg, h_{atm} = 765 mmHg.
- Step 2: Find the absolute pressure height head: $h_{abs} = h_{atm} - h_{vac}$ = 765 - 620 = 145 mmHg.
- Step 3: Convert the absolute head to pressure using hydrostatic formula: $P_{abs} = \rho_{Hg} * g * h_{abs}$.
- Step 4: Compute: P_{abs} = 13560 kg/m^3 * 9.81 m/s^2 * 0.145 m = 19,288 Pa = 19.29 kPa.

Numerical Problem 3: U-Tube Manometer I

A U-tube manometer is used to measure the pressure of water in a pipeline. The left limb is open to the atmosphere, and the right limb is connected to the pipe. The mercury level in the left limb is 15 cm higher than the level in the right limb, and the water-mercury interface is in the right limb. Determine the gauge and absolute pressure in the pipe. Take local atmospheric pressure as 101.3 kPa.

- Step 1: Write the pressure balance equation at the horizontal datum of the interface: $P_{pipe} + \rho_w * g * h_w = P_{atm} + \rho_m * g * h_m$.
- Step 2: Since interface is at right limb, mercury column of height 15 cm is in left limb. Thus, $P_{gauge} = P_{pipe} - P_{atm} = (\rho_m - \rho_w) * g * h = (13600 - 1000) * 9.81 * 0.15$.
- Step 3: Solve for gauge pressure: $P_{gauge} = 12600 * 9.81 * 0.15 = 18{,}540.9$ Pa $= 18.54$ kPa.

- Step 4: Calculate absolute pressure: $\mathrm{P_{abs}} = \mathrm{P_{atm}} + \mathrm{P_{gauge}} = 101.3\ \mathrm{kPa} + 18.54\ \mathrm{kPa} = 119.84\ \mathrm{kPa}$.

Engineering Applications and Summary I

Atmospheric and absolute pressure measurements form the cornerstone of hydrodynamic analyses, particularly in studying suction lifts in pumps, NPSH calculations, and cavitation phenomena. In hydraulic machinery like centrifugal pumps and turbines, local pressure can drop below vapor pressure, leading to cavitation. Engineers must always distinguish between absolute and gauge pressures to prevent structural damage and design errors in fluid systems.

- Always convert gauge pressures to absolute pressures before using thermodynamic state equations or gas laws.
- Local atmospheric pressure decreases with altitude, affecting pump suction capability (suction lift decreases at higher altitudes).
- Cavitation occurs when local absolute pressure falls to the vapor pressure of the fluid at the operating temperature.

Engineering Applications and Summary II

- Regular calibration of pressure measuring instruments (barometers, Bourdon gauges, transducers) is vital for accurate engineering data.

Concept of Total pressure and center of pressure (No derivation) on a vertical immersed body I

Course: Fluid Mechanics and Hydraulic Machinery

- Analysis of hydrostatic forces acting on vertical immersed planes.
- Differentiating between total pressure (force) and center of pressure (point of application).
- Introduction to pressure measurement devices including manometers and mechanical gauges.
- Application of hydrostatic principles to solve numerical manometer problems.

Concept of Total Pressure (F) I

Total pressure is the force exerted by a static fluid on an immersed surface when the fluid comes in contact with it. For a plane vertical surface, the pressure increases linearly with depth due to gravity. The total pressure force (F) is calculated as the product of the pressure at the centroid of the area and the total area of the surface. Mathematically: $F = \rho * g * \bar{h} * A$, where ρ is the fluid density, g is the acceleration due to gravity, A is the surface area, and $\bar{h}$ is the depth of the centroid (center of gravity) of the surface from the free liquid surface.

- Total pressure acts normal to the immersed surface at every point.
- For a static fluid, shear stress is zero, hence hydrostatic force is always normal.
- Centroid depth ($\bar{h}$) depends strictly on the geometry and depth of immersion of the body.

Concept of Total Pressure (F) II

- Total pressure is measured in Newtons (N) or Kilonewtons (kN).

Concept of Center of Pressure (h*) I

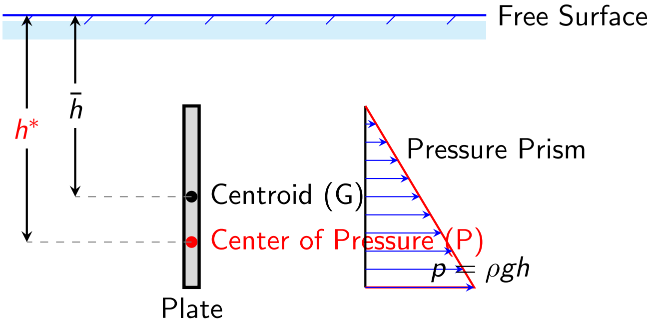
The center of pressure is the point of application of the total pressure force on the immersed surface. Because hydrostatic pressure increases linearly with depth, the pressure distribution across a vertical surface is non-uniform (triangular or trapezoidal). Therefore, the total pressure force acts at a point lower than the centroid (center of gravity) of the surface. The depth of the center of pressure (h*) from the free surface is given by: $h^* = (I_G / (A * \bar{h})) + \bar{h}$, where I_G is the moment of inertia of the area about its centroidal axis parallel to the free liquid surface.

- Center of pressure (h*) is always below the centroid ($\bar{h}$) for vertical and inclined surfaces.
- For a horizontal surface, pressure is uniform and the center of pressure coincides with the centroid.
- As the depth of immersion increases, the term $(I_G / (A * \bar{h}))$ decreases, bringing h* closer to $\bar{h}$.

Concept of Center of Pressure (h*) II

- The formula for h* is presented without derivation as per curriculum requirements.

Diagram: Hydrostatic Force on Vertical Immersed Plate I



- Visual representation of a vertical plane surface submerged in liquid.
- The centroid (G) is the geometric center, whereas the center of pressure (P) lies lower.

Diagram: Hydrostatic Force on Vertical Immersed Plate II

- Hydrostatic pressure increases linearly with depth, resulting in a triangular/trapezoidal force distribution.
- The total pressure force (F) acts horizontally through the center of pressure (P).

Geometric Properties of Immersed Shapes I

To apply the formula h* = (I_G / (A * $\bar{h}$)) + $\bar{h}$, we must compute the area (A), centroid ($\bar{h}$), and centroidal moment of inertia (I_G) for common shapes. For a Rectangle (width b, depth d): A = b*d, Centroid is at d/2, I_G = b*(d^3)/12. For a Circle (diameter d): A = π*(d^2)/4, Centroid is at d/2, I_G = π*(d^4)/64. For a Triangle (base b, height h): A = 0.5*b*h, Centroid is at h/3 from base, I_G = b*(h^3)/36. For a Trapezoid (base a, top b, height h): A = (a+b)*h/2, I_G = (a^2 + 4ab + b^2)*h^3 / (36*(a+b)).

- Centroid ($\bar{h}$) must always be measured from the free liquid surface, not the top of the shape.
- I_G must be taken about the axis passing through the centroid and parallel to the free liquid surface.
- Moment of inertia represents the resistance of the shape's area to rotation about the centroidal axis.
- Units of Area: m^2; Units of Moment of Inertia: m^4.

Introduction to Pressure Measuring Devices I

Pressure measuring devices are critical for monitoring fluid systems. They are categorized based on their working principles: 1) Manometers: Use hydrostatic columns of liquid to measure pressure differences. They include Piezometers, U-Tube Manometers, Single Column Manometers, and Differential Manometers. 2) Mechanical Gauges: Use elastic members that deform under pressure, with the deformation converted to dial movement. They include Bourdon Tube Gauges, Diaphragm Gauges, Bellows Gauges, and Dead-weight Testers.

- Manometers are highly accurate and simple, containing no moving mechanical parts.
- A Piezometer is the simplest manometer, but can only measure positive gauge pressures of liquids.
- Differential manometers are used to measure pressure drops across valves, orifices, and venturimeters.

- Mechanical gauges are suitable for high pressures, vacuums, dynamic loads, and industrial environments.

Simple U-Tube Manometer: Working Principle I

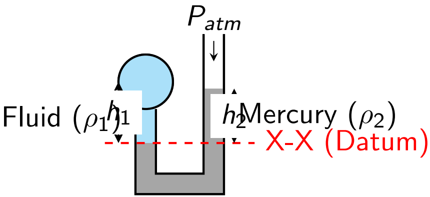
A U-tube manometer consists of a glass tube bent in a U-shape, partially filled with a high-density liquid (usually mercury, sp. gr. 13.6). One limb is connected to the point where pressure is to be measured, and the other is open to the atmosphere. The pressure of the fluid pushes the manometric liquid down in the connected limb and up in the open limb. To analyze, we establish a horizontal datum line at the interface of the two fluids in the lower limb. The pressure at this datum level in the left limb must equal the pressure at the same level in the right limb: $P_{pipe} + \rho_1 * g * h_1 = \rho_2 * g * h_2$.

- The manometric fluid must be immiscible with the fluid in the pipe and have a higher density.
- A horizontal line in a continuous static fluid of the same type is an isobar (line of equal pressure).

Simple U-Tube Manometer: Working Principle II

- Gauge pressure is calculated assuming atmospheric pressure is zero.
- If the pressure is below atmospheric (vacuum), the mercury level in the connected limb rises.

Diagram: Simple U-Tube Manometer I



- Schematic showing positive gauge pressure measurement in a fluid line.
- The datum line (X-X) represents the boundary above which pressure balance is computed.
- Left limb pressure at datum: $\mathrm{P_A} + \rho_1 * \mathrm{g} * \mathrm{h}_1$.
- Right limb pressure at datum: $\mathrm{P_{atm}} + \rho_2 * \mathrm{g} * \mathrm{h}_2$ (where $\mathrm{P_{atm}}$ is 0 for gauge pressure).

Numerical Problem 1: Gauge Pressure in Water Pipeline I

Problem: A simple U-tube manometer containing mercury (specific gravity 13.6) is connected to a pipe containing water (density 1000 kg/m^3). The mercury level in the right limb (open to atmosphere) is 20 cm above the pipe center, and the interface in the left limb is 15 cm below the pipe center. Find the pressure in the pipe in kPa. Solution: 1) Identify given parameters: Density of water $\rho_1 = 1000$ kg/m^3, Density of mercury $\rho_2 = 13.6 * 1000 = 13600$ kg/m^3. Height of water above datum $h_1 = 15$ cm $= 0.15$ m. Height of mercury above datum $h_2 = 15$ cm $+ 20$ cm $= 35$ cm $= 0.35$ m. 2) Apply hydrostatic equilibrium: $P_{pipe} + \rho_1 * g * h_1 = \rho_2 * g * h_2$. 3) Compute: $P_{pipe} + (1000 * 9.81 * 0.15) = 13600 * 9.81 * 0.35$ =¿ $P_{pipe} + 1471.5 = 46695.6$ =¿ $P_{pipe} = 45224.1$ Pa $= 45.22$ kPa.

- Convert all physical dimensions to SI units (meters) before calculations.

Numerical Problem 1: Gauge Pressure in Water Pipeline II

- Verify the height h_2 contains the total column of mercury above the datum line.
- The datum line is always established at the liquid interface in the lower limb.
- Final pressure $P_{pipe} = 45.22$ kPa (gauge).

Numerical Problem 2: Differential U-Tube Manometer I

Problem: A differential U-tube manometer contains mercury (sp. gr. 13.6) and is connected to two pipes A and B. Pipe A contains oil (sp. gr. 0.9) at a pressure of 180 kPa. Pipe B contains water. The center of pipe B is 1.2 m below the center of A. The mercury level in the left limb (A) is 0.6 m below the center of A, and in the right limb (B) is 0.4 m below the center of B. Find the pressure in pipe B. Solution: 1) Convert values: $\rho_{\text{oil}} = 900$ kg/m^3, $\rho_{\text{water}} = 1000$ kg/m^3, $\rho_{\text{Hg}} = 13600$ kg/m^3. $P_A = 180{,}000$ Pa. 2) Set datum at the lower interface (left limb, 0.6 m below A). Right interface is 0.4 m below B. Since B is 1.2 m below A, the right interface is 1.2 + 0.4 = 1.6 m below A. The mercury column height is 1.6 - 0.6 = 1.0 m. 3) Equate pressures at datum: $P_A + \rho_{\text{oil}}*g*h_{\text{oil}} = P_B + \rho_{\text{water}}*g*h_{\text{water}} + \rho_{\text{Hg}}*g*h_{\text{Hg}}$. Here, $h_{\text{oil}} = 0.6$ m, $h_{\text{water}} = 0.4$ m, $h_{\text{Hg}} = 1.0$ m. 4) Calculate: $180{,}000 + 900*9.81*0.6 = P_B +$

Numerical Problem 2: Differential U-Tube Manometer II

1000*9.81*0.4 + 13600*9.81*1.0 =¿ 185,297.4 = P_B + 3,924 + 133,416 =¿ P_B = 185,297.4 - 137,340 = 47,957.4 Pa = 47.96 kPa.

- Differential manometers measure the difference in pressure between two pipes.
- We must carefully trace the heights of fluid columns above the chosen datum line.
- Applying pressure balance: $P_{leftdatum} = P_{rightdatum}$.
- Pressure in Pipe B is computed to be 47.96 kPa (gauge).

Course: Fluid Mechanics and Hydraulic Machinery

- Instructor: Professor of Mechanical Engineering
- Topic: Fluid Kinematics & Flow Visualization
- Mathematical and Physical Formulations of Streamlines
- Application to Steady and Unsteady Flow Fields

Definition and Core Concept of Streamlines I

A streamline is an imaginary line drawn through a flowing fluid such that the tangent to it at any point gives the direction of the velocity vector of the fluid particle at that point. Because the velocity vector represents the direction of motion, there can be no flow across a streamline at any instant. In steady flow, the pattern of streamlines remains constant over time, meaning the path of an individual fluid particle coincides exactly with the streamline passing through its initial position.

- Tangent at any point represents the local velocity vector direction.
- Zero normal velocity component across a streamline: $u_n = 0$.
- Streamline patterns are instantaneous snapshots of flow direction.
- For steady flows, streamlines, pathlines, and streaklines are identical.

Mathematical Formulation in Two-Dimensional Flow I

Mathematically, the equation of a streamline is derived from the condition that the velocity vector is parallel to the differential line element along the streamline. In a two-dimensional Cartesian coordinate system, the velocity vector is V = u*i + v*j and the differential displacement is dr = dx*i + dy*j. The cross product of V and dr must be zero, yielding the differential equation: dx/u = dy/v. Integrating this equation for a given velocity field yields the families of curves representing the streamlines.

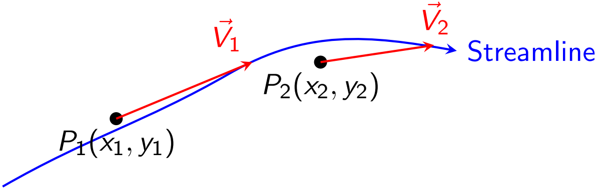
- Derived from parallel vectors condition: V x dr = 0.
- Differential equation for 2D flow: dx/u = dy/v.
- Slope of the streamline is given by dy/dx = v/u.
- Integrating the relation gives the stream function constant curves, psi(x,y) = constant.

Streamlines in Three-Dimensional Flow I

For three-dimensional fluid flows, the velocity vector V is defined as u*i + v*j + w*k, and the differential path element dr is dx*i + dy*j + dz*k. The collinearity condition V x dr = 0 yields a system of three differential equations: $dx/u = dy/v = dz/w$. Solving these simultaneous differential equations defines the 3D space curves of the streamlines. These curves are the intersection of two families of surfaces, often described using stream surfaces.

- 3D streamline differential relation: $dx/u = dy/v = dz/w$.
- u, v, and w are the velocity components in the x, y, and z directions respectively.
- Represents a system of ordinary differential equations (ODEs).
- Streamlines in 3D cannot cross each other except at stagnation points.

Vector Representation of Streamlines I



- Graphical representation of tangent vectors.
- Velocity vector V is tangent to the streamline curve at all points.
- Instantaneous velocity components u and v determine the local slope.
- Visualizing laminar flow path alignment.

Important Properties of Streamlines I

Streamlines possess several key mathematical and physical properties that govern their behavior in fluid kinematics. First, streamlines cannot cross each other because a single point in space cannot have two different velocity vectors at the same instant (except at stagnation points where velocity is zero). Second, fluid cannot cross a streamline by definition. Third, the spacing between streamlines indicates the flow velocity: closely spaced streamlines indicate high-velocity regions (flow convergence), whereas widely spaced streamlines indicate low-velocity regions (flow divergence) due to mass conservation.

- Streamlines never intersect, except at stagnation points ($V = 0$).
- No mass flow across a streamline; they act as physical boundaries.

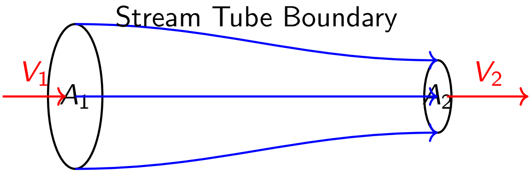
Important Properties of Streamlines II

- Proximity of streamlines corresponds to velocity magnitude (density is proportional to speed).
- Streamline curvature indicates presence of pressure gradients normal to the flow.

The Concept of a Stream Tube I

A stream tube is a tubular region of fluid bounded by a bundle of streamlines passing through a closed curve in the flow field. Since streamlines cannot be crossed by definition, no fluid can enter or leave the sides of a stream tube. Consequently, fluid can only enter and exit through the open ends of the tube. For steady, incompressible flow, the volumetric flow rate remains constant along any cross-section of the stream tube, satisfying the continuity equation: $A1 * V1 = A2 * V2$.

- A stream tube is formed by a closed loop of streamlines.
- Zero lateral mass transfer across the boundaries of a stream tube.
- Acts like an imaginary solid pipe or duct within the fluid.
- Governed by the continuity equation: $A_1\ V_1 = A_2\ V_2$ for incompressible flow.



- Converging stream tube showing velocity acceleration.
- Area reduction from A1 to A2 results in increased velocity (V2 ¿ V1).
- Continuous streamline walls prevent lateral mass transport.
- Visual representation of the 1D continuity model.

Relation to the Stream Function I

For two-dimensional, incompressible flow, streamlines can be represented mathematically using a scalar function called the stream function, denoted by psi(x,y). The velocity components are defined in terms of the stream function as u = d(psi)/dy and v = -d(psi)/dx. Along any streamline, the stream function remains constant (d(psi) = 0). The difference between the stream function values of two streamlines is equal to the volumetric flow rate per unit depth passing between those two streamlines: Q = psi_2 - psi_1.

- Stream function psi(x, y) exists for 2D incompressible flows.
- Velocity components: u = partial(psi)/partial(y), v = -partial(psi)/partial(x).
- Line of constant stream function (psi = constant) is a streamline.
- Flow rate between two streamlines: dQ = psi_2 - psi_1.

Streamlines vs. Pathlines vs. Streaklines I

In fluid kinematics, three types of flow lines are defined: streamlines, pathlines, and streaklines. A streamline is an instantaneous line tangent to velocity vectors. A pathline is the actual trajectory traced by a single fluid particle over a period of time (Lagrangian description). A streakline is the locus of all fluid particles that have passed through a specific common injection point (e.g., smoke or dye injection). While these three lines differ significantly in unsteady flows, they are identical in steady flows because the velocity field is time-invariant.

- Streamline: Instantaneous velocity direction map (Eulerian perspective).
- Pathline: Individual particle history over time (Lagrangian perspective).
- Streakline: Continuous injection marker line (experimental visualization).

Streamlines vs. Pathlines vs. Streaklines II

- All three lines coincide perfectly in steady flow fields.

Streak line I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 10: Streak Lines - Definition, Formulation, and Application

- Core kinematic concept in fluid flow visualization
- Distinction from streamlines and pathlines
- Key tool for experimental fluid mechanics

Physical Concept of a Streak Line I

A streakline is the locus of all fluid particles that have passed sequentially through a prescribed common point in space. It is a snapshot of the locations of these particles at a single instant in time. This is commonly observed in experimental fluid mechanics where dye, smoke, or helium bubbles are continuously injected into a flow field at a fixed spatial location (the source).

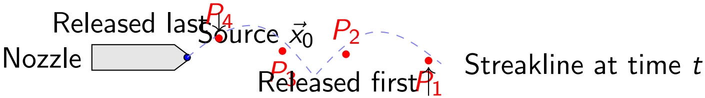
- Represents a continuous injection of dye/smoke from a fixed point.
- Formed by a succession of particles passing through the same injection point.
- Crucial for flow visualization in experimental setups.

Mathematical Formulation I

Mathematically, let a particle be identified by the time t_0 at which it was released from the injection source located at $\vec{x}_0 = (x_0, y_0, z_0)$. The position of this particle at any later time t is given by the pathline equation $\vec{x}_p(t; \vec{x}_0, t_0)$. For a fixed current time t and a fixed injection point $\vec{x}_0$, the streakline is the curve parameterized by the release time t_0, where $t_0 \leq t$.

- Injection point is fixed: $\vec{x}_0 = \text{const}$.
- Observation time is fixed: $t = \text{const}$.
- Parameterizing variable is the release time $t_0 \in [-\infty, t]$.

Streak Line Generation Diagram I



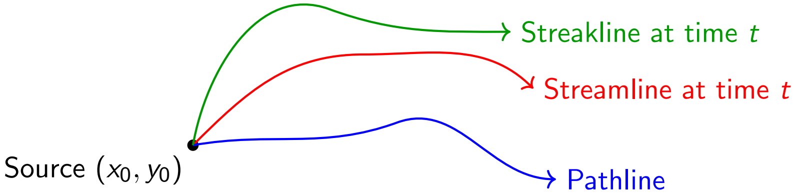
- Dye source injected at $\vec{x}_0$ continuously.
- Particles P_1, P_2, P_3, P_4 are released at successive times $t_1 < t_2 < t_3 < t_4$.
- At instant t, particle P_1 has traveled furthest downstream.

Steady vs. Unsteady Flow Relations I

In a steady flow field, the velocity vector at any point in space remains constant with time, i.e., $\partial\vec{V}/\partial t = 0$. Under steady state conditions, streamlines, pathlines, and streaklines are completely identical. However, in unsteady flow, where the velocity field changes with time, these three lines diverge and represent different physical trajectories.

- Steady Flow: Streamline = Pathline = Streakline.
- Unsteady Flow: Streamline $\neq$ Pathline $\neq$ Streakline.
- The identity in steady flow simplifies engineering flow visualization interpretation.

Flow Lines Divergence in Unsteady Flow I



- Streamlines show velocity directions at a single instant t.
- Pathlines track the history of a single particle over time.
- Streaklines connect all particles passing through the source up to time t.

Analytical Example: Unsteady Velocity Field I

To analytically differentiate the lines, consider the 2D velocity field defined by $u = kx$, $v = ky(1 + at)$ where k and a are constants. While the streamline at any instant is found by integrating $dy/dx = v/u = y(1 + at)/x$, the pathline is found by solving the ODEs $dx/dt = kx$ and $dy/dt = ky(1 + at)$. The streakline at time t is then obtained by mapping the positions of particles launched from (x_0, y_0) at all times $\tau \leq t$.

- Allows quantitative verification of divergence.
- Streamline equation at time t: $\ln(y) = (1 + at)\ln(x) + C$.
- Pathline depends on the dynamic history of the velocity field.

Experimental Visualization Techniques I

In fluid dynamics laboratories, streaklines are generated using physical markers. Common techniques include: (1) Smoke wires or smoke generators in wind tunnels, where kerosene oil is evaporated on a heated wire; (2) Dye injection (such as fluorescein or food coloring) in water channels; (3) Hydrogen bubble generation, where electrolysis of water at a fine cathode wire creates tiny hydrogen bubbles that serve as tracers.

- Smoke wires generate thin streak sheets in gas flows.
- Dye injection requires neutral density to avoid buoyancy effects.
- Hydrogen bubbles provide both qualitative and quantitative velocity data.

Application in Hydraulic Machinery I

Streaklines are vital for diagnosing flow behavior in turbomachinery. In a centrifugal pump or Francis turbine, flow is highly unsteady due to rotor-stator interactions. By visualizing streaklines, engineers can identify flow separation zones, recirculating eddies, and wake propagation behind guide vanes or runner blades. This visual diagnostic guides blade shape optimization to minimize hydrodynamic losses.

- Visualizes wakes and vortex shedding behind turbine blades.
- Identifies energy-dissipating recirculation zones.
- Aids in validating Computational Fluid Dynamics (CFD) models.

Key Takeaways: Streak lines I

Streak lines are an essential concept in kinematics of fluids, representing a snapshot of particles injected from a single point. In steady flow, streamlines, pathlines, and streaklines are identical, whereas in unsteady flow, they differ. Understanding these differences prevents errors in interpreting flow visualizations from wind tunnels, water channels, and turbomachinery diagnostics.

- A streakline is a snapshot of all particles from a single source.
- They coincide with streamlines and pathlines only in steady flows.
- Critical tool for experimental fluid dynamics and turbine design.

Path line and Stream tube I

Course: Fluid Mechanics and Hydraulic Machinery Instructor: Professor of Mechanical Engineering This lecture covers the fundamental kinematic descriptions of fluid motion, specifically pathlines, stream tubes, and various classifications of fluid flows.

- Kinematics of Fluid Flow
- Pathline vs. Streamline vs. Streakline
- Concept of a Stream Tube
- Classification of Fluid Flows

Fluid Kinematics: Describing Motion I

Fluid kinematics deals with the geometry of motion and translation of fluid particles without considering the forces causing the motion. To visualize and analyze flow fields, we define specific line trajectories: streamlines, pathlines, and streaklines. While they differ conceptually, they coincide under steady flow conditions.

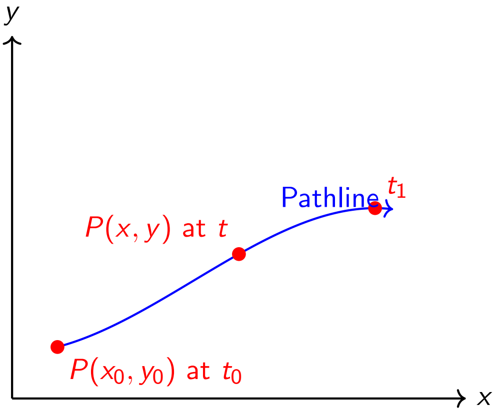
- Lagrangian description tracks individual fluid particles over time, which directly relates to the concept of a pathline.
- Eulerian description focuses on fixed points in space, defining velocity fields and leading to streamlines.
- A pathline is the actual trajectory traced by a single, specific fluid particle over a period of time.
- In steady flow, velocity at any spatial point is invariant with time, meaning streamlines, pathlines, and streaklines are identical.

The Pathline: Definition and Governing Equations I

A pathline represents the history of a fluid particle's locations. Mathematically, it is obtained by integrating the velocity field over time for a marked particle. If a particle is at position (x0, y0, z0) at time t0, its pathline coordinates (x, y, z) at any time t are solved using the kinematic relationship $dx/dt = u$, $dy/dt = v$, and $dz/dt = w$.

- Governing differential equations: $dx/dt = u(x,y,z,t)$, $dy/dt = v(x,y,z,t)$, $dz/dt = w(x,y,z,t)$ with initial conditions $x(t0)=x0$, $y(t0)=y0$, $z(t0)=z0$.
- Pathlines can intersect themselves at different times, unlike streamlines which cannot intersect (except at stagnation points) at a single instant.
- Experimentally visualized by taking a long-exposure photograph of a single glowing or marked tracer particle.

Pathline Visualization I



- The curve shows the trajectory of a single particle starting at P(x0,y0) at t0.

Pathline Visualization II

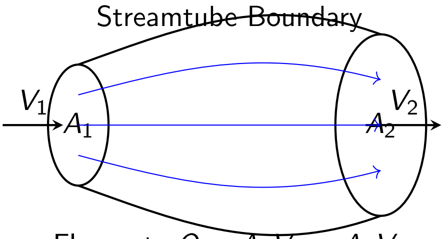
- Coordinates are tracked dynamically through time-dependent integrations.
- Points along the curve represent locations at sequential times t0 ¡ t ¡ t1.

Concept of a Stream Tube I

A stream tube is a tube-like surface formed by drawing streamlines through every point on a closed curve in a fluid flow. Because streamlines are everywhere tangent to the velocity vector, there is no velocity component normal to the surface of the stream tube. Consequently, fluid cannot cross the lateral boundaries of a stream tube.

- A stream tube behaves like an impermeable, solid pipe of varying cross-sectional area, even though its walls are made of fluid streamlines.
- For steady, incompressible flow, the volumetric flow rate ($Q = A1*V1 = A2*V2$) remains constant along the stream tube.
- This conservation of mass within a stream tube forms the foundation for deriving the one-dimensional continuity equation.

Stream Tube Structure and Flow Conservation I



Flow rate $Q = A_1 V_1 = A_2 V_2$

- No flow crosses the boundary surface because velocity vectors are tangent to the streamlines.
- Cross-sectional area increases from A1 to A2, corresponding to a decrease in average velocity from V1 to V2 in incompressible flow.
- Stream tubes are useful for simplifying three-dimensional flows into equivalent one-dimensional analysis.

Types of Fluid Flow: Steady vs. Unsteady & Uniform vs. Non-uniform I

Fluid flows are classified based on how flow parameters change with respect to time and space. Steady flow occurs when parameters (velocity, pressure, density) at any point do not change with time ($dq/dt = 0$). Uniform flow occurs when velocity does not change in magnitude and direction from point to point along the flow path at any given instant ($dV/ds = 0$).

- Steady flow: Velocity vector $V = V(x,y,z)$ is independent of time t. Example: Flow in a pipe at a constant discharge rate.
- Unsteady flow: $V = V(x,y,z,t)$ varies with time. Example: Flow in a pipe when a valve is being opened or closed.
- Uniform flow: Velocity gradient with respect to space s is zero. Example: Flow through a straight pipe of constant cross-section.

Types of Fluid Flow: Steady vs. Unsteady & Uniform vs. Non-uniform II

- Non-uniform flow: Velocity changes from point to point at any instant. Example: Flow through a diverging or converging pipe.

Types of Fluid Flow: Laminar vs. Turbulent & Rotational vs. Irrotational I

Flows are also classified based on the nature of particle trajectories and fluid rotation. In laminar flow, fluid particles move in well-defined paths or layers (laminae) with no macroscopic mixing. In turbulent flow, fluid particles move in highly irregular, chaotic paths causing intense mixing. Rotational flow occurs when fluid particles rotate about their own axes, whereas irrotational flow features zero particle rotation.

- Laminar flow: Occurs at low velocities and high viscosities (low Reynolds numbers). Flow of highly viscous oil is typically laminar.
- Turbulent flow: Occurs at high velocities and low viscosities (high Reynolds numbers). Most engineering applications are turbulent.

Types of Fluid Flow: Laminar vs. Turbulent & Rotational vs. Irrotational II

- Rotational flow: Curl of velocity vector is non-zero (vorticity $\vec{\omega} = \nabla \times \vec{V} \neq 0$). Characterized by shear deformation.
- Irrotational flow: $\vec{\omega} = \nabla \times \vec{V} = 0$. Potential flow theory applies, allowing the use of a velocity potential function.

Types of Fluid Flow: Compressible vs. Incompressible & Dimensionality I

Compressible flow experiences significant density variations throughout the flow field (d_{rho} $!= 0$), typically relevant when flow velocities approach or exceed the speed of sound (Mach number M ¿ 0.3). Incompressible flow maintains a constant density ($d_{rho} = 0$). Additionally, flow is classified as one-, two-, or three-dimensional depending on the number of spatial coordinates needed to describe the velocity field.

- Incompressible flow: Density is assumed constant. Water flows and low-speed airflows (M ¡ 0.3) are treated as incompressible.
- Compressible flow: Density changes are significant. High-speed aerodynamics and gas dynamics fall into this category.
- One-Dimensional (1D) Flow: Velocity is a function of only one space coordinate, e.g., $V = u\hat{i}$. Flow in a narrow stream tube is often modeled as 1D.

Types of Fluid Flow: Compressible vs. Incompressible & Dimensionality II

- Two-Dimensional (2D) Flow: $V = u\hat{i} + v\hat{j}$, flow over an infinite wing. Three-Dimensional (3D) Flow: $V = u\hat{i} + v\hat{j} + w\hat{k}$, real-world complex flows.

Lecture Summary: Key Takeaways I

Understanding flow lines and flow classifications is critical for establishing appropriate boundary conditions and selecting governing equations in fluid machinery design. Pathlines track individual histories, streamlines represent instantaneous velocity directions, and stream tubes conserve mass. Proper flow classification ensures correct simplification of Navier-Stokes equations.

- Pathline relation: $dx/u = dy/v = dz/w = dt$ (integrated to find coordinates).
- Stream tube mass conservation: mass flow rate = rho * A * V = constant (steady, 1D flow).
- Flow classification directly determines whether to use viscous or inviscid, compressible or incompressible, and steady or unsteady solver formulations in CFD.

Lecture Summary: Key Takeaways II

- Review Questions: Differentiate between a streamline and a pathline. Why can fluid not cross stream tube boundaries?

Continuity & Momentum Equation I

Course: Fluid Mechanics and Hydraulic Machinery

- Topic 1: Conservation of Mass
- Topic 2: Conservation of Linear Momentum
- Topic 3: Conservation of Angular Momentum

System vs. Control Volume & Reynolds Transport Theorem (RTT) I

In fluid mechanics, analyzing a fixed mass of fluid (a system) is often impractical due to fluid motion. Instead, we use a Control Volume (CV) which is a defined region in space through which fluid flows. The mathematical bridge between system-based conservation laws (Lagrangian description) and control volume analysis (Eulerian description) is the Reynolds Transport Theorem (RTT). For an extensive property B and intensive property $b = B/m$, RTT is expressed as: $d/dt(B_{sys}) = d/dt \int_{CV} b\ \rho\ dV + \int_{CS} b\ \rho\ (V_r \cdot n)\ dA$, where V_r is the relative velocity.

- Lagrangian formulation tracks a fixed identity fluid system.
- Eulerian formulation focuses on flow through a fixed spatial control volume.

System vs. Control Volume & Reynolds Transport Theorem (RTT) II

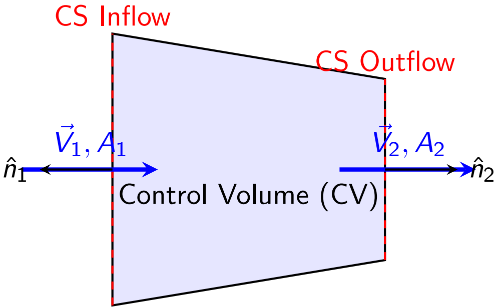
- Reynolds Transport Theorem relates the rate of change of a system property to the rate of change within the CV plus the net flux across the Control Surface (CS).
- Intensive property 'b' equals mass for conservation of mass, and velocity vector for momentum.

The Continuity Equation (Conservation of Mass) I

Applying RTT with $B = m$ (mass) and $b = 1$ yields the integral form of the Continuity Equation: $d/dt \int_{CV} \rho\, dV + \int_{CS} \rho\, (V_r \cdot n)\, dA = 0$. For steady, incompressible flow with one-dimensional inlets and outlets, the equation simplifies to the conservation of volumetric flow rate: $\sum A_{in} V_{in} = \sum A_{out} V_{out}$, or $Q = A_1 V_1 = A_2 V_2$. This states that fluid velocity is inversely proportional to cross-sectional area.

- Mass cannot be created or destroyed within a non-relativistic system.
- For steady flow, the rate of mass accumulation within the control volume is zero.
- Incompressible flow assumes constant density (ρ = constant), decoupling mass conservation from thermodynamics.
- Area-averaged velocity V is defined as $(1/A) \int V\, dA$ across the flow cross-section.

Control Volume Diagram for Continuity I



- Illustration of a 1D control volume with one inlet and one outlet.
- V1 and V2 represent average flow velocities across cross-sections A1 and A2.
- Normal vectors n1 and n2 point outward from the control surface.

- For steady flow, the net mass flow rate (inflow minus outflow) must equal zero.

Differential Form of the Continuity Equation I

Applying the Divergence Theorem to the control surface integral of the continuity equation yields the differential form: $\partial \rho / \partial t + \nabla \cdot (\rho V) = 0$. In Cartesian coordinates, this expands to: $\partial \rho / \partial t + \partial (\rho u)/\partial x + \partial (\rho v)/\partial y + \partial (\rho w)/\partial z = 0$. This form represents conservation of mass at an infinitesimal point in space, which is critical for computational fluid dynamics (CFD) and local flow analysis.

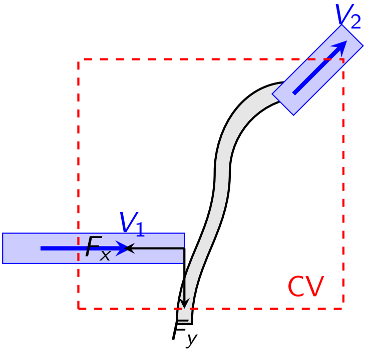
- The term $\partial \rho / \partial t$ represents local rate of density change.
- The divergence term $\nabla \cdot (\rho V)$ represents net mass efflux per unit volume.
- For steady, incompressible flows, the differential continuity equation reduces to $\nabla \cdot V = 0$.
- Satisfying continuity is a prerequisite for solving the Navier-Stokes equations.

The Linear Momentum Equation I

Applying Newton's second law ($F = ma$) to a fluid system using RTT, where $B = m\ V$ (momentum) and $b = V$, yields the Linear Momentum Equation: $\sum F = d/dt \int_{CV} V\ \rho\ dV + \int_{CS} V\ \rho\ (V_r \cdot n)\ dA$. The net force $\sum F$ acting on the control volume includes body forces (gravity, electromagnetic) and surface forces (pressure, shear stress, and reaction forces from solid walls).

- Momentum is a vector quantity; the equation must be resolved into x, y, and z components.
- Body forces act on the entire mass of the control volume (e.g., $W = \int_{CV} \rho\ g\ dV$).
- Surface forces act directly on the boundaries of the control volume.
- Atmospheric pressure acts uniformly on all surfaces and cancels out if gauge pressure is used.

Control Volume Diagram for Momentum I



- Schematic of a high-speed fluid jet striking a stationary curved vane.
- The fluid is deflected by angle theta, changing its momentum vector.

Control Volume Diagram for Momentum II

- Fx and Fy are the reaction forces required to hold the vane in place.
- The control volume (dashed box) encloses the vane and the fluid jet.

Analysis of Force on a Pipe Bend I

A classic application of the momentum equation is calculating the force exerted by a fluid flowing through a pipe bend. Since the flow direction changes, the fluid exerts a dynamic reaction force on the pipe wall. The equation in the x-direction is: P_1 A_1 - P_2 A_2 $\cos(\theta)$ - $F_x = \rho$ Q (V_2 $\cos(\theta)$ - V_1), where F_x is the force exerted by the pipe on the fluid. The structural support must withstand the equal and opposite reaction force.

- Pressure forces act normal to the control surfaces at the inlet and outlet (P_1 A_1 and P_2 A_2).
- The term ρ Q (V_2 $\cos(\theta)$ - V_1) represents the rate of momentum change in the x-direction.
- A similar equation is written for the y-direction to find the total force magnitude and direction.
- The pipe bend must be anchored with a thrust block to prevent structural failure due to dynamic forces.

Angular Momentum Equation (Moment of Momentum) I

For rotating hydraulic machinery like pumps and turbines, torque and angular momentum are the primary variables of interest. The Moment of Momentum equation is derived by taking the cross product of the position vector r with the terms in Newton's second law: $\sum T = d/dt \int_{CV} (r \times V)\, \rho\, dV + \int_{CS} (r \times V)\, \rho\, (V_r \cdot n)\, dA$. Under steady conditions with uniform inlets/outlets, this yields Euler's Turbine Equation: $T = \rho\, Q\, (r_2\, V_{t2} - r_1\, V_{t1})$, where V_t is the tangential velocity component.

- This equation is fundamental to the design of turbomachinery (turbines, pumps, compressors).
- Torque T is equal to the net rate of angular momentum efflux through the control surface.
- Euler's Turbine Equation relates torque directly to fluid density, flow rate, and tangential velocity change.

Angular Momentum Equation (Moment of Momentum) II

- Power generated or absorbed is given by $P = T\,\omega$, where ω is the rotational speed of the impeller.

Practical Engineering Applications & Summary I

The Continuity and Momentum equations form the bedrock of hydraulic engineering. They enable the analysis of jet propulsion, force on aerodynamic bodies, thrust blocks for water mains, water hammer phenomena, and the performance of hydraulic machines like Pelton wheels, Francis turbines, and centrifugal pumps. Understanding control volume boundaries, choosing appropriate reference frames, and identifying all acting forces are critical steps in successful engineering design.

- Continuity ensures mass conservation; momentum conservation yields forces and torques.
- Selecting the correct control volume (fixed, moving, or deforming) simplifies the analysis.
- Combining these equations with Bernoulli's equation allows complete hydrodynamic system design.

Practical Engineering Applications & Summary II

- CFD software packages solve these equations in differential form over millions of grid cells.

Bernoulli's theorem and its derivation from energy and Euler's equation I

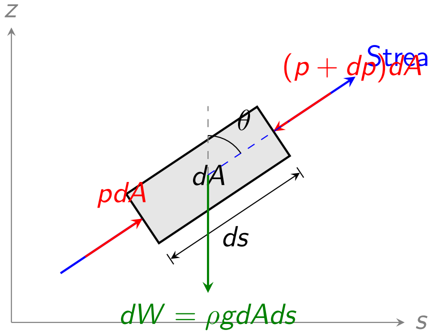
Course: Fluid Mechanics and Hydraulic Machinery

- Fundamental energy conservation principles in fluid flow
- Derivation of Bernoulli's equation from Euler's equation of motion
- Alternative derivation based on the Work-Energy Theorem
- Key assumptions, physical interpretations, and engineering applications

Introduction and Physical Significance I

Bernoulli's equation is a statement of the conservation of mechanical energy for a flowing fluid. It states that along a streamline of a steady, incompressible, and frictionless flow, the total energy per unit mass—comprising pressure energy, kinetic energy, and potential energy—remains constant. This provides a simple yet powerful relationship between velocity, pressure, and elevation.

- Total Head (H) is constant at all points along a streamline.
- Represents the balance between static pressure, dynamic pressure, and hydrostatic pressure.
- Can be derived by integrating Euler's equation of motion along a streamline.
- Units can be expressed as energy per unit volume (N/m^2 or Pa), energy per unit mass (J/kg), or head (m of fluid column).



- A cylindrical fluid element of cross-sectional area dA and length ds is considered along a streamline.
- Pressure force pdA acts in the direction of flow; $(p + dp)dA$ acts against the flow direction.

Diagram: Forces on a Fluid Element II

- The gravity force $dW = \rho g dA ds$ acts vertically downwards, opposing motion by its component $dW \cos\theta$.
- The angle θ is between the upward vertical (z-axis) and the streamline (s-direction).

Fundamental Assumptions I

The derivation of the classical Bernoulli's equation depends on key simplifying assumptions. While real-world fluid flows are complex, these assumptions allow for a very accurate approximation in many practical engineering designs.

- Steady Flow: The velocity, density, and pressure at any point in the flow field do not change over time.
- Incompressible Flow: The fluid density (ρ) is constant throughout the flow field (typically valid for liquids and low-speed gas flows).
- Inviscid (Frictionless) Flow: Shear stresses due to viscosity are assumed to be zero, meaning no mechanical energy is lost to heat.
- Streamline Flow: The relationship holds along any single streamline, although it holds everywhere in an irrotational flow.

Euler's Equation of Motion I

Euler's equation is Newton's Second Law ($F = ma$) applied to a fluid flow where viscosity is neglected. By summing all forces acting along the streamline direction (s-direction) on a fluid element of mass $dm = \rho dAds$, we establish a differential relation for pressure, velocity, and elevation.

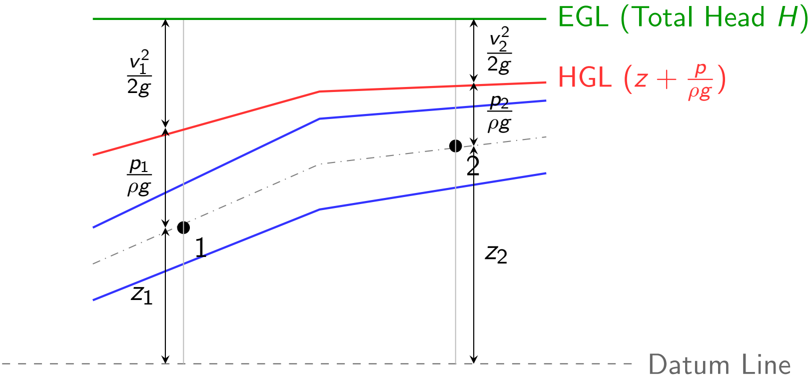
- Sum of forces along s: $F_s = pdA - (p + dp)dA - dW \cos\theta = -dpdA - \rho gdAds \cos\theta$.
- Since $\cos\theta = dz/ds$, the gravity term becomes $-\rho gdAdz$.
- Acceleration along the streamline: $a_s = v\frac{\partial v}{\partial s} + \frac{\partial v}{\partial t}$. Under steady conditions, this simplifies to $v\frac{dv}{ds}$.
- Equating forces to mass times acceleration: $-dpdA - \rho gdAdz = (\rho dAds)v\frac{dv}{ds}$, which simplifies to $\frac{dp}{\rho} + vdv + gdz = 0$.

Derivation: Integrating Euler's Equation I

Integrating Euler's equation along a streamline yields Bernoulli's equation. Since density (ρ) and gravity (g) are constant, each differential term can be integrated directly between any two points along a streamline.

- Integration of Euler's equation: $\int \frac{dp}{\rho} + \int v dv + \int g dz = \text{constant}.$
- For incompressible flow ($\rho = \text{constant}$), we get: $\frac{p}{\rho} + \frac{v^2}{2} + gz = \text{constant}$ (Energy per unit mass).
- Dividing by g yields: $\frac{p}{\rho g} + \frac{v^2}{2g} + z = \text{constant}$ (Energy per unit weight, or 'Head' form).
- This shows that elevation head (z), pressure head ($\frac{p}{\rho g}$), and velocity head ($\frac{v^2}{2g}$) sum to a constant value along a streamline.

Diagram: Energy Heads in Pipe Flow I



- Elevation Head (z) represents the potential energy relative to a reference datum line.
- Pressure Head ($\frac{p}{\rho g}$) represents the potential energy due to static fluid pressure.

Diagram: Energy Heads in Pipe Flow II

- Velocity Head ($\frac{v^2}{2g}$) represents the kinetic energy of the fluid per unit weight.
- Energy Grade Line (EGL) shows total head, while Hydraulic Grade Line (HGL) shows piezometric head ($z + \frac{p}{\rho g}$).

Derivation from Conservation of Energy I

Bernoulli's equation can also be derived by applying the Work-Energy Theorem to a steady fluid flow through a streamtube. The work done on a fluid element by pressure forces at the inlet and outlet equals the net change in the element's kinetic and potential energies as it moves along the flow.

- Work done by pressure at inlet: $W_1 = p_1 A_1 ds_1 = p_1 dV$. Work done at outlet: $W_2 = -p_2 A_2 ds_2 = -p_2 dV$.
- Net work done on the fluid: $W_{\text{net}} = (p_1 - p_2)dV = (p_1 - p_2)\frac{dm}{\rho}$.
- Change in kinetic energy: $d(\text{KE}) = \frac{1}{2}dm(v_2^2 - v_1^2)$. Change in potential energy: $d(\text{PE}) = dmg(z_2 - z_1)$.
- Equating net work to energy change ($W_{\text{net}} = d(\text{KE}) + d(\text{PE})$) and dividing by dm yields: $\frac{p_1}{\rho} + \frac{v_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{v_2^2}{2} + gz_2$.

Real-World Limitations and Losses I

In actual hydraulic machinery and piping systems, fluids are real and possess viscosity, leading to friction and heat dissipation. Engineers modify the idealized Bernoulli's equation to account for head loss (h_L) and mechanical work transfer (h_{pump} or h_{turbine}).

- Real fluid equation:
  $\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 + h_{\text{pump}} = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 + h_L + h_{\text{turbine}}$.
- Head Loss (h_L) consists of major loss (pipe wall friction) and minor losses (valves, fittings, expansions).
- Kinetic Energy Correction Factor (α) is introduced to account for non-uniform velocity distribution: $\alpha \frac{v^2}{2g}$.
- For compressible flows (gases), thermodynamic relations (e.g., isothermal or adiabatic) must replace the constant-density assumption.

Bernoulli's theorem is the core principle behind many flow measurement devices, aerodynamic phenomena, and hydraulic systems. By manipulating cross-sectional areas, engineers control fluid velocity and pressure to perform useful work.

- Venturi Meter: Measures discharge by creating a pressure drop at a constricted throat, calibrated via $Q = C_d \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}} \sqrt{2g\Delta h}$.
- Orifice Meter: A simpler, cheaper flow measurement device utilizing a sharp-edged orifice plate.
- Pitot-Static Tube: Measures local velocity at a point by combining static and stagnation pressure ports: $v = \sqrt{2g\Delta h}$.
- Aerodynamic Lift: Airfoils generate lift due to velocity and pressure differences between upper and lower surfaces.

Limitations in Fluid Mechanics and Hydraulic Machinery I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 14: Analysis of Theoretical Assumptions, Operational Boundaries, and Hydro-Mechanical Constraints

- Inherent limitations of simplified theoretical models (Bernoulli, Potential Flow).
- Physical and mathematical constraints in boundary layer theory and separation.
- Operational limits of empirical pipe flow equations and dynamic similarity models.
- Mechanical and hydrodynamic operational envelopes of hydraulic turbines and pumps.
- Severe transient limits including water hammer and vapor column separation.

Limitations of Bernoulli's Equation I

Bernoulli's equation is a simplified energy conservation relation derived along a streamline. In real engineering systems, several physical phenomena restrict its direct applicability, requiring correction factors or advanced conservation equations.

- Frictionless Flow: Neglects shear stresses due to dynamic viscosity, leading to underestimation of pressure drop (corrected using the Darcy-Weisbach head loss term).
- Incompressible Flow: Assumes constant fluid density; invalid for gas flows where the Mach number exceeds 0.3, where thermodynamic equations of state must be coupled.
- Steady State Assumption: Fails to account for transient changes in velocity, such as the water hammer phenomenon in pipes where dV/dt is large.

Limitations of Bernoulli's Equation II

- No Energy Addition/Extraction: Cannot cross boundary regions where shaft work (turbines, pumps) or heat transfer occurs without explicit work/heat terms.

Potential Flow Theory & D'Alembert's Paradox I

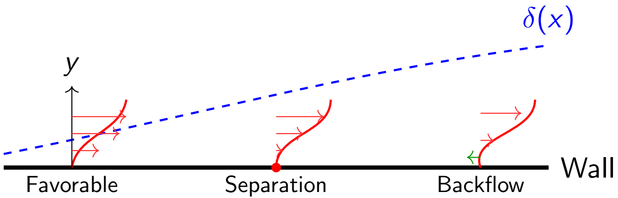
Potential flow theory assumes inviscid, irrotational flow, allowing the velocity field to be represented as the gradient of a scalar velocity potential. While mathematically elegant, it fails to predict aerodynamic drag on arbitrary bodies.

- D'Alembert's Paradox: For an inviscid fluid flowing past a closed body, the net drag force is mathematically zero, which contradicts physical reality.
- Slip Boundary Condition: Assumes fluid slips freely at solid boundaries, ignoring the physical no-slip condition that creates shear stress and boundary layers.
- Flow Separation Failure: Cannot predict boundary layer detachment or wake formation, which are the primary sources of pressure drag (form drag) in bluff bodies.

Potential Flow Theory & D'Alembert's Paradox II

- Inability to Predict Lift Limits: Cannot model aerodynamic stall or boundary-layer-driven circulation limits without artificial additions like the Kutta condition.

Boundary Layer Theory Limitations I



- Prandtl's boundary layer equations assume that viscous effects are confined to a thin layer near the wall.
- Limitation: When subjected to an adverse pressure gradient (dp/dx ¿ 0), the boundary layer decelerates, leading to flow separation.
- Mathematical Singularity: Standard boundary layer equations exhibit a singularity at the separation point, making downstream calculations invalid without coupled viscous-inviscid interaction or Navier-Stokes solvers.

Boundary Layer Theory Limitations II

- Low Reynolds Number Limits: Fails at low Reynolds numbers (Re ¡ 100) where the boundary layer thickness is comparable to the characteristic body length.

Limitations of Empirical Pipe Flow Formulas I

While the Darcy-Weisbach equation is physically derived, engineers often use empirical formulas like Hazen-Williams or Manning's equations for convenience. These empirical relationships have strict operational limits.

- Hazen-Williams Limits: Valid ONLY for water at normal temperatures (about 15-25°C). It completely ignores fluid viscosity changes and is inaccurate for cold or hot water.
- Manning's Equation: Designed specifically for open-channel flows and gravity-driven pipes. It becomes inaccurate in high-velocity pressurized flows.
- Moody Chart Transition Zone: In the transition region between laminar and fully turbulent flow (2000 ¡ Re ¡ 4000), flow is highly unstable, and friction factors fluctuate unpredictably.

Limitations of Empirical Pipe Flow Formulas II

- Non-Newtonian Fluids: None of these standard formulas apply to non-Newtonian fluids (like slurries or polymers), which require rheological model modifications.

Similitude & Scale Effects in Model Testing I

Physical model testing relies on geometric, kinematic, and dynamic similarity. However, satisfying multiple dynamic similarity criteria simultaneously is often physically impossible, leading to 'scale effects'.

- The Reynolds-Froude Conflict: In open-channel or ship model testing, matching both Froude number (Fr) and Reynolds number (Re) requires a model fluid with a kinematic viscosity that scale-down rules cannot satisfy using water.
- Compromise in Testing: Models are usually scaled based on the dominant force (e.g., Froude for wave resistance), while accepting errors in the other (Reynolds-based viscous drag), requiring numerical corrections.

Similitude & Scale Effects in Model Testing II

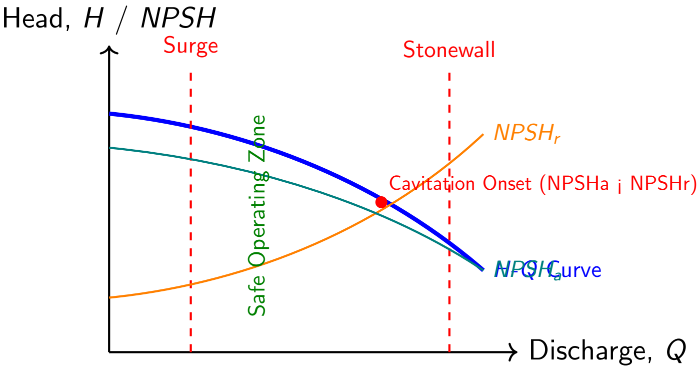
- Surface Tension Effects: Small scale models suffer from disproportionately high Weber number (We) effects, where capillary forces alter free-surface behaviors that are negligible in the prototype.
- Boundary Layer Scaling: Turbulent boundary layers do not scale linearly, resulting in thicker relative boundary layers in models compared to full-scale prototypes.

Operational Limits of Hydraulic Turbines I

Hydraulic turbines cannot operate across arbitrary head and discharge ranges. Mechanical constraints, hydro-elastic vibrations, and phase changes dictate strict operating envelopes.

- Part-Load Operation: Operating Francis or Kaplan turbines at low discharge (¡40
- Cavitation Limit: Occurs when local pressure drops below the fluid's vapor pressure (p_v), forming vapor bubbles that collapse violently, eroding runner blades and dropping efficiency.
- Runaway Speed: If the load is suddenly lost, the turbine accelerates to its maximum speed (runaway speed), posing catastrophic centrifugal stress risks to the rotor.
- Thoma's Cavitation Factor (σ): Must be kept above the critical cavitation factor (σ_c) by controlling the installation height (z_s) relative to the tailrace level.

Pump Operational Envelope and NPSH Limits I



- Centrifugal pumps must operate within a window defined by aerodynamic/hydraulic instability on the left (Surge) and choking/cavitation on the right (Stonewall).

Pump Operational Envelope and NPSH Limits II

- Net Positive Suction Head Required ($NPSH_r$) is a pump characteristic that increases quadratically with flow rate due to internal friction and acceleration.
- Net Positive Suction Head Available ($NPSH_a$) is a system characteristic that decreases with flow rate due to frictional head losses in the suction piping.
- Limitation: When $NPSH_a < NPSH_r$, the pressure at the impeller eye falls below the vapor pressure, leading to cavitation, mechanical damage, and sudden head drop.

Centrifugal vs. Positive Displacement Limits I

Dynamic pumps (centrifugal) and positive displacement (reciprocating/rotary) pumps have diametrically opposite performance limitations, dictating their application boundaries.

- Viscosity Constraints: Centrifugal pumps experience severe efficiency degradation at kinematic viscosities above 100 cSt, whereas positive displacement pumps operate efficiently with high-viscosity fluids.
- Head-Flow Characteristics: Centrifugal pumps cannot deliver high heads at very low flow rates without entering the surge region; positive displacement pumps deliver constant flow independent of head.
- Priming Requirement: Centrifugal pumps cannot evacuate air from the suction line (cannot prime themselves) due to low density of air; positive displacement pumps are self-priming.

Centrifugal vs. Positive Displacement Limits II

- Pressure Pulsation: Positive displacement pumps suffer from cyclic pressure pulsations, requiring pulsation dampeners to prevent fatigue in piping systems.

Transient Flow Limits & Water Hammer I

Steady-state pipe flow analysis fails during sudden valve closure or pump failure, leading to water hammer transients governed by Joukowsky's equation. Preventing these limits is a critical safety parameter.

- Joukowsky Equation: $\Delta p = \rho c \Delta V$, where c is the speed of the acoustic pressure wave in the fluid. Fast valve closure ($t < 2L/c$) generates extreme pressure peaks.
- Elastic Limits: The speed of sound c is limited by the bulk modulus of the fluid and the elasticity (Young's modulus) of the pipe walls, which dampens the wave.
- Column Separation: If the transient pressure falls below vapor pressure, local vaporization occurs, followed by catastrophic shock waves when the vapor pocket collapses.

Transient Flow Limits & Water Hammer II

- Control Limits: Mitigation requires surge tanks, air release valves, or slow-closing bypass valves, which introduce complexity and capital cost to piping design.

Lecture 15: Applications of Hydraulic Machinery I

Course: Fluid Mechanics and Hydraulic Machinery Instructor: Professor of Mechanical Engineering Topic: Practical Industrial Applications of Fluid Power Systems and Hydraulic Machines.

- Introduction to fluid power transfer systems
- Classification of hydraulic devices based on energy conversion
- Industrial significance of hydrostatic and hydrodynamic machines

Hydrostatic vs. Hydrodynamic Systems I

Hydraulic systems are fundamentally categorized into hydrostatic systems, which utilize high-pressure fluid at low velocities to transmit power, and hydrodynamic systems, which leverage the kinetic energy of fluid flow at lower pressures. Hydrostatic devices operate on Pascal's law, where pressure applied to a confined fluid is transmitted undiminished in all directions, whereas hydrodynamic devices rely on momentum transfer principles governed by Euler's turbine equation.

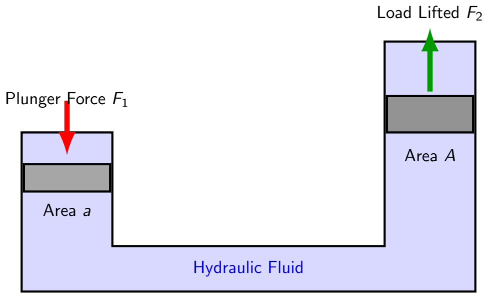
- Hydrostatic systems: Hydraulic Press, Jack, Accumulator, Intensifier, Crane.
- Hydrodynamic systems: Hydraulic Coupling, Torque Converter, Hydraulic Ram.
- Force transmission vs. kinetic energy transfer mechanisms.
- Typical operating pressures range from 100 to 1000 bar for hydrostatic systems.

The Hydraulic Press: Principles and Operation I

The hydraulic press is an industrial machine that utilizes Pascal's principle to generate large compressive forces. It consists of two main cylinders of different cross-sectional areas connected by a pipe filled with hydraulic fluid. A small force applied to a smaller plunger creates a hydrostatic pressure, which is transmitted to a larger ram, producing a magnified lifting or pressing force proportional to the ratio of their cross-sectional areas.

- Mechanical Advantage: Ratio of ram area (A) to plunger area (a), given by $MA = A/a$.
- Work conservation dictates that the stroke distance of the ram is smaller than the plunger by the factor a/A.
- Typical applications include metal forging, extrusion, deep drawing, and powder compaction.
- High-pressure seals are critical to prevent volumetric losses and maintain pressure stability.

Schematic Diagram of a Hydraulic Press I



$$\boxed{\frac{F_1}{a} = \frac{F_2}{A} \implies F_2 = F_1\left(\frac{A}{a}\right)}$$

- Graphical representation of hydrostatic multiplication of force.
- Left cylinder of cross-sectional area 'a' houses the plunger acting under input force 'F1'.
- Right cylinder of cross-sectional area 'A' houses the ram lifting the output load 'F2'.

Schematic Diagram of a Hydraulic Press II

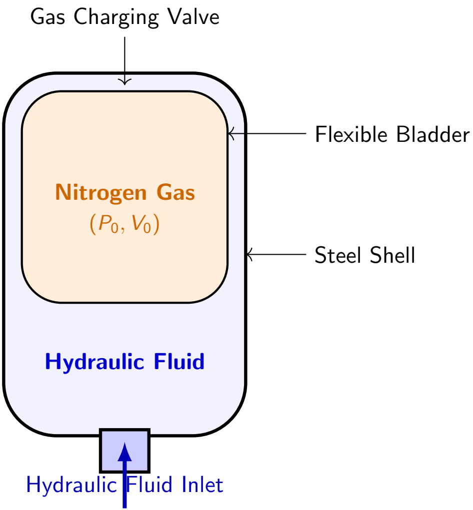
- System uses liquid media to transfer force directly via pressure containment.

Hydraulic Accumulator: Energy Storage Device I

A hydraulic accumulator is a pressure storage reservoir in which a non-compressible hydraulic fluid is held under pressure by an external source. This external source can be a spring, a raised weight, or a compressed gas. Its primary functions are to store energy to cope with peak demands, smooth out pulsations from reciprocating pumps, and act as a dampener to absorb hydraulic shock (water hammer) in fluid circuits.

- Types: Dead-weight (gravity-loaded), Spring-loaded, and Gas-charged (bladder/piston) accumulators.
- Gas-charged accumulators follow the polytropic gas law ($P*V^n = C$) during charging and discharging.
- Stores energy during periods of low demand and releases it rapidly during peak flow requirements.
- Enhances system efficiency by allowing the use of a smaller capacity primary hydraulic pump.

Schematic of a Gas-Charged Bladder Accumulator I



- Schematic of a gas-filled bladder type hydraulic accumulator.

Schematic of a Gas-Charged Bladder Accumulator II

- The bladder separates the nitrogen gas from the system hydraulic fluid, preventing gas dissolution.
- As fluid pressure exceeds gas pre-charge pressure, fluid enters the accumulator, compressing the gas.
- Stored potential energy of the compressed gas is returned to the fluid circuit during system pressure drop.

Hydraulic Intensifier: Pressure Multiplication I

A hydraulic intensifier is a hydrostatic device used to increase the intensity of fluid pressure by leveraging differential area pistons. It consists of a low-pressure cylinder containing a sliding piston connected to a smaller high-pressure plunger. Fluid supplied at a moderate pressure to the larger piston area produces a significantly higher pressure in the fluid displaced by the smaller plunger, satisfying the force equilibrium.

- Governing Equation: $P_{high} = P_{low} * (A / a)$, where A is the large area and a is the small area.
- Used when the main pump pressure is insufficient for specific local actuators (e.g., intensifier on a press).
- Essential in ultra-high-pressure applications like water jet cutting (pressures up to 4000-6000 bar).
- Continuous intensifiers employ control valves and dual pistons to achieve steady output flow.

Hydraulic Jacks and Cranes: Heavy Lifting Systems I

Hydraulic jacks and cranes are classic mechanical systems designed to lift heavy loads using hydrostatic force. A hydraulic jack is portable and operates on a manual or motorized pump cylinder with non-return valves to raise the piston incrementally. A hydraulic crane utilizes a larger hydraulic cylinder, a jib, and a system of pulleys to magnify the stroke distance, allowing large vertical lifting heights from a relatively short cylinder stroke.

- Hydraulic Jack: Uses suction and delivery check valves to prevent backflow and load dropping.
- Mechanical advantage is multiplied by the lever arm ratio of the manual pumping handle.
- Hydraulic Crane: Employs a set of movable and fixed pulleys to multiply stroke distance.
- Safety relief valves and counter-balance valves are integrated to prevent collapse under load.

Hydrodynamic Devices: Hydraulic Coupling I

A hydraulic coupling (or fluid coupling) is a hydrodynamic device used to transmit rotating mechanical power without mechanical contact. It consists of an impeller (pump) driven by the input shaft and a runner (turbine) connected to the output shaft, both enclosed in a housing filled with low-viscosity hydraulic oil. The impeller accelerates the fluid radially outward, and this high-velocity fluid impinges on the runner blades, transferring momentum and driving the output shaft.

- Slip (s): The difference between input speed (N1) and output speed (N2), expressed as s = (N1 - N2)/N1.
- Efficiency is directly related to slip: n = (N2 / N1) = 1 - s.
- Allows smooth start-up of heavy machinery, protecting the motor from shock loads and stalls.
- Since there is no mechanical connection, there is no wear and tear on the torque transmitting elements.

Hydraulic Torque Converter: Torque Multiplication I

A hydraulic torque converter is an advancement over the fluid coupling that not only transmits power but also multiplies the input torque. It achieves this by introducing a stationary third element called the stator (or guide vanes) between the impeller and the runner. The stator redirects the oil returning from the runner so that it assists the impeller's rotation, thereby increasing the momentum transfer and multiplying torque, especially at high slip conditions.

- Stator action: Curve-redirecting vanes multiply torque up to 2 to 3 times the engine input.
- Torque ratio is maximum at stall (output speed $= 0$) and decreases as runner speed approaches impeller speed.
- Used extensively in automatic transmissions of heavy vehicles, locomotives, and earthmoving machinery.

- Lock-up clutches are often integrated to eliminate slip losses and improve efficiency at high cruising speeds.

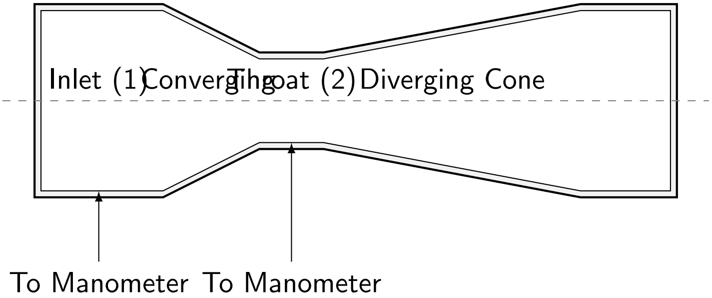
Course: Fluid Mechanics and Hydraulic Machinery

- Topic: Fluid Flow Measurement
- Lecture 16: Venturimeter and Orifice Meter Principles
- Focus: Working mechanisms, equations, comparison, and numerical applications

Introduction to Venturimeter I

A Venturimeter is a flow measurement device that operates on Bernoulli's principle. By contracting the flow through a converging cone, it converts pressure energy into kinetic energy, creating a measurable pressure differential between the inlet and the throat.

- Used for measuring discharge in high-pressure and high-velocity pipelines.
- Consists of three main parts: Converging cone (20 to 22 degrees), Throat, and Diverging cone (5 to 7 degrees).
- The small angle of the diverging cone minimizes boundary layer separation and flow turbulence.



- Section 1 (Inlet) has diameter d_1, area a_1, and pressure p_1.
- Section 2 (Throat) has diameter d_2, area a_2, and pressure p_2.
- Pressure taps at Section 1 and Section 2 connect to a differential U-tube manometer.

Discharge Equations and Coefficient of Discharge I

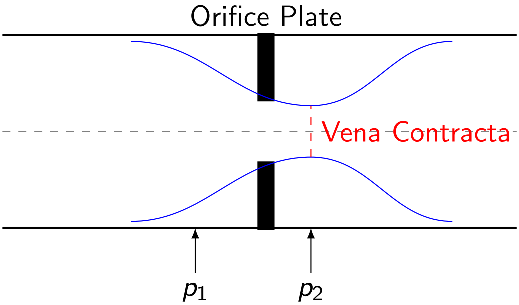
By applying Bernoulli's equation and the continuity equation between the inlet and throat, we derive the theoretical and actual discharge equations. The coefficient of discharge accounts for viscous losses.

- Theoretical discharge: $Q_{th} = \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$, where h is the differential pressure head.
- Actual discharge: $Q_{act} = C_d \times Q_{th} = C_d \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$.
- Coefficient of Discharge (C_d) typically ranges from 0.95 to 0.99 for a standard Venturimeter.
- For a differential manometer with heavier liquid (sp. gr. S_m) and fluid (sp. gr. S_f): $h = x(\frac{S_m}{S_f} - 1)$.

Orifice Meter: Principle of Operation I

An Orifice Meter is a thin, flat plate with a sharp-edged circular hole (orifice) installed concentrically in a pipe. It acts as a sudden restriction, forcing the flow to contract and create a measurable pressure difference.

- Works on the same flow contraction principle as a Venturimeter but is much more compact and inexpensive.
- The flow contracts downstream of the plate, reaching its minimum area at the 'Vena Contracta'.
- Discharge equation: $Q_{act} = C_d \frac{a_0 a_1 \sqrt{2gh}}{\sqrt{a_1^2 - a_0^2}}$, where a_0 is the orifice area.
- Coefficient of Discharge (C_d) is low, usually around 0.60 to 0.65, due to significant eddy losses.



- The orifice plate blocks the path, leaving only the central circular opening.
- The Vena Contracta forms slightly downstream (approx. 0.5 times the pipe diameter) where streamlines are parallel.
- High turbulence and flow separation cause high permanent pressure loss.

Venturimeter vs. Orifice Meter: Comparison I

Engineers choose between the Venturimeter and the Orifice Meter based on a trade-off between installation cost, accuracy requirements, piping space constraints, and energy efficiency.

- Accuracy & Discharge Coefficient: Venturimeter is highly accurate ($C_d \approx 0.98$); Orifice meter is less accurate ($C_d \approx 0.62$).
- Permanent Head Loss: Low in Venturimeter (streamlined flow); very high in Orifice meter (sudden expansion and eddies).
- Cost & Installation: Venturimeter is expensive and bulky; Orifice meter is cheap, simple, and takes minimal space.
- Maintenance: Orifice plate edges are prone to wear and erosion, changing C_d over time; Venturimeter is highly durable.

Numerical Problem 1: Horizontal Venturimeter I

A horizontal venturimeter with inlet diameter 20 cm and throat diameter 10 cm measures water flow. The inlet pressure is 17.658 N/cm^2 and the throat vacuum pressure is 30 cm of mercury. Find the discharge. Take Cd = 0.98.

- Inlet Area $a_1 = \pi/4 \times 0.2^2 = 0.031416 \text{ m}^2$, Throat Area $a_2 = \pi/4 \times 0.1^2 = 0.007854 \text{ m}^2$.
- Inlet Pressure Head:
  $p_1/\rho g = 17.658 \times 10^4/(1000 \times 9.81) = 18$ m of water.
- Throat Vacuum Head:
  $p_2/\rho g = -30$ cm of Hg $= -0.3 \times 13.6 = -4.08$ m of water.
- Differential Head
  $h = p_1/\rho g - p_2/\rho g = 18 - (-4.08) = 22.08$ m of water.
- Discharge
  $Q = 0.98 \times \frac{0.031416 \times 0.007854 \times \sqrt{2 \times 9.81 \times 22.08}}{\sqrt{0.031416^2 - 0.007854^2}} = 0.1655 \text{ m}^3/\text{s}$ (165.5 L/s).

Numerical Problem 2: Inclined Venturimeter I

An inclined venturimeter (30 degrees to horizontal) with inlet diameter 30 cm and throat diameter 15 cm carries oil of sp. gr. 0.8. A differential mercury manometer shows a reading of 20 cm. If Cd = 0.98, find the discharge.

- Areas: $a_1 = 0.070686$ m^2 and $a_2 = 0.017671$ m^2.
- Differential head
  $h = x(\frac{S_m}{S_o} - 1) = 0.20(\frac{13.6}{0.8} - 1) = 3.2$ m of oil.
- The angle of inclination does not affect the differential head calculation when using a differential manometer.
- Using the discharge equation: $Q = C_d \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$.
- Discharge
  $Q = 0.98 \times \frac{0.070686 \times 0.017671 \times \sqrt{2 \times 9.81 \times 3.2}}{\sqrt{0.070686^2 - 0.017671^2}} = 0.1417$ m^3/s (141.7 L/s).

Numerical Problem 3: Orifice Meter Calibration I

An orifice meter with orifice diameter 15 cm is inserted in a 30 cm diameter pipe carrying water. Pressure gauges at the inlet and throat read 14.7 N/cm^2 and 9.81 N/cm^2 respectively. Find the discharge if Cd = 0.6.

- Inlet Area $a_1 = 0.070686$ m^2, Orifice Area $a_0 = 0.017671$ m^2.
- Differential pressure
  $\Delta p = 14.7 - 9.81 = 4.89$ N/cm^2 $= 4.89 \times 10^4$ N/m^2.
- Differential pressure head
  $h = \Delta p/\rho g = 4.89 \times 10^4/(1000 \times 9.81) = 4.985$ m of water.
- Using the orifice discharge equation: $Q = C_d \frac{a_1 a_0 \sqrt{2gh}}{\sqrt{a_1^2 - a_0^2}}$.
- Discharge
  $Q = 0.6 \times \frac{0.070686 \times 0.017671 \times \sqrt{2 \times 9.81 \times 4.985}}{\sqrt{0.070686^2 - 0.017671^2}} = 0.1083$ m^3/s
  (108.3 L/s).

Course: Fluid Mechanics and Hydraulic Machinery

- Introduction to local flow velocity measurement instruments.
- Theoretical foundations and Bernoulli's equation application.
- Design, operation, and practical limitations of Pitot and Pitot-static probes.

Operating Principle of the Pitot Tube I

The Pitot tube is a classic differential pressure instrument used to measure local fluid velocity at a specific point in a flow field. Invented by Henri Pitot in 1732, its operation relies on the concept of flow stagnation. When a tube with an open end is aligned parallel to the flow direction facing upstream, the incoming fluid is forced to decelerate to zero velocity at the tube inlet. This point of zero velocity is called the stagnation point. At this point, the kinetic energy of the fluid is entirely converted into potential energy, causing a localized pressure rise known as the stagnation pressure.

- Measures point velocity, unlike a Venturimeter which measures average volumetric flow rate.
- Inlet opening faces directly into the flow to capture total pressure.
- Brings fluid to rest isentropically (without friction or heat transfer) at the tip.

Operating Principle of the Pitot Tube II

- Stagnation pressure is higher than the static pressure of the surrounding flowing fluid.

Theoretical Derivation using Bernoulli's Equation I

To derive the velocity equation, Bernoulli's equation is applied along a streamline from an upstream point (1) to the stagnation point (2) inside the Pitot tube. Assuming steady, incompressible, and frictionless flow along a horizontal streamline: $P1/\gamma + (V1^2)/(2g) + Z1 = P2/\gamma + (V2^2)/(2g) + Z2$. Since the streamline is horizontal, $Z1 = Z2$. Since point (2) is a stagnation point, $V2 = 0$. The equation simplifies to $P2/\gamma = P1/\gamma + (V1^2)/(2g)$. Solving for velocity V1 yields the theoretical velocity: $V1 = \text{sqrt}(2 * (P2 - P1) / \rho)$, where P2 is stagnation pressure, P1 is static pressure, and ρ is fluid density.

- $P1/\gamma$ represents static pressure head (potential energy of flowing fluid).
- $(V1^2)/(2g)$ represents dynamic pressure head (kinetic energy of flowing fluid).

Theoretical Derivation using Bernoulli's Equation II

- P2/γ represents total/stagnation pressure head (sum of static and dynamic head).
- The difference (P2 - P1) is the dynamic pressure, directly proportional to velocity squared.

Actual Velocity and Coefficient of Velocity I

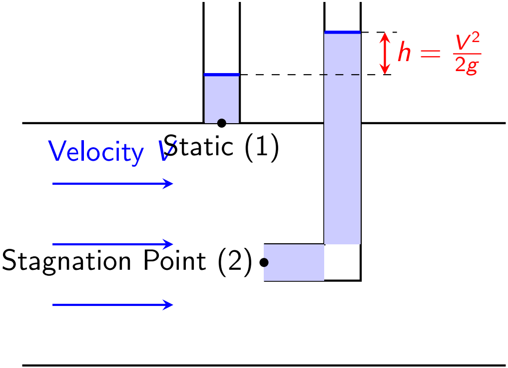
In real fluid flows, viscous effects, flow turbulence, and instrument misalignment introduce energy losses, which causes the actual velocity to be slightly lower than the theoretical value derived from the idealized Bernoulli's equation. To correct for these losses, a dimensionless calibration factor called the Coefficient of Velocity (Cv) is introduced. The actual velocity is calculated as $V_{actual} = Cv * sqrt(2 * g * h)$, where h is the difference in height of the liquid columns in a differential manometer, $h = (P2 - P1) / (\rho_{fluid} * g)$ or $h = y * (\rho_{mano}/\rho_{fluid} - 1)$ depending on the manometer configuration.

- Coefficient of Velocity (Cv) typically ranges between 0.98 and 1.00 for well-designed tubes.
- Cv is determined experimentally through calibration in a known flow field.

Actual Velocity and Coefficient of Velocity II

- Frictional effects along the outer wall of the probe are the main source of Cv deviation from unity.
- Accurate pressure measurement using U-tube manometers or digital transducers is crucial.

Pitot Tube Schematic in Flow I



- Point 1 (Static) measures only static pressure using a wall tap perpendicular to flow.
- Point 2 (Stagnation) measures total pressure (static + dynamic) at the tube entrance.

Pitot Tube Schematic in Flow II

- The difference in fluid height (h) in the columns represents the dynamic head.
- This configuration shows the separation of static and stagnation pressure measurement lines.

A complete physical understanding of fluid flow velocity measurements requires distinguishing between three types of pressure. Static pressure (P_{static}) is the thermodynamic pressure of the fluid, representing molecular random motion, measured perpendicular to the flow to avoid kinetic interference. Dynamic pressure ($P_{dynamic} = 0.5 * \rho * V^2$) represents the kinetic energy of the fluid per unit volume. Stagnation pressure ($P_{stagnation}$) is the pressure the fluid would attain if brought to rest without loss (isentropically). $P_{stagnation}$ is always the sum of P_{static} and $P_{dynamic}$, and represents the maximum possible pressure in the flow line.

- Static pressure is measured parallel to the velocity vector through flush wall taps.
- Dynamic pressure cannot be measured directly; it must be calculated as P_{total} - P_{static}.

Understanding Static, Dynamic, and Total Pressures II

- Stagnation pressure is measured by placing a port directly perpendicular to the oncoming velocity vector.
- For compressible flows (Mach ¿ 0.3), dynamic pressure calculations must include compressibility correction factors.

The Pitot-Static Tube (Prandtl Tube) I

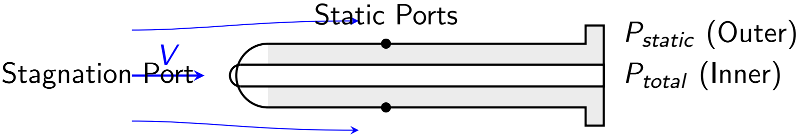
To simplify velocity measurements and eliminate the need for separate wall taps, the Pitot-static tube (or Prandtl tube) combines both static and stagnation pressure measurement ports into a single, coaxial probe. The instrument consists of two concentric tubes: the inner tube opens at the nose to face the flow, capturing stagnation pressure, while the outer tube has small radial holes on its sides to measure static pressure. Because both ports are located on the same instrument, the static and stagnation pressures are measured at virtually the same spatial location in the flow field, minimizing errors due to velocity gradients.

- Coaxial design transmits both pressures through separate chambers to a differential manometer.
- Static ports are placed sufficiently far back from the nose (typically 8 tube diameters) to avoid flow disruption.

The Pitot-Static Tube (Prandtl Tube) II

- Highly portable instrument, easily inserted into ducts, wind tunnels, and pipes.
- Essential for flight velocity measurements where wall-mounted taps are impractical.

Pitot-Static Tube Construction I



- Inner channel carries the total stagnation pressure from the nose tip.
- Concentric outer chamber communicates static pressure from the lateral holes.
- Aerodynamically shaped nose reduces separation and preserves streamline alignment.
- Output lines lead to differential pressure transducers for electronic logging.

Practical Applications of Pitot Tubes I

Pitot and Pitot-static tubes are vital instrumentation tools across numerous industries. In aerospace engineering, they form the cornerstone of aircraft air data systems, providing airspeed inputs to pilot displays and flight control computers. In wind tunnel testing, they map velocity profiles around aerodynamic models. In industrial HVAC and process engineering, they are used to audit ventilation airflow rates, calibrate commercial flow meters, and measure exhaust velocities in stacks and flues. Due to their simple geometry, they introduce negligible pressure drop (no permanent head loss) compared to orifice plates or venturimeters.

- Aerospace: Measures airspeed, Mach number, and indirectly contributes to altitude calculations.
- Aerodynamics: Critical for automotive aerodynamic testing and Formula 1 telemetry.

Practical Applications of Pitot Tubes II

- HVAC Systems: Used for balancing airflow in large commercial air-handling ducts.
- Minimal flow obstruction makes them ideal for temporary velocity diagnostic audits.

Limitations and Troubleshooting in Operation I

Despite their reliability, Pitot tubes suffer from limitations that must be addressed in practice. First, they are point-measurement devices, meaning they only capture velocity at a single coordinate. To determine volumetric flow rate, a traverse (multiple readings at different radial positions) must be performed. Second, they are highly sensitive to alignment; yaw angles greater than 5 degrees introduce significant errors. Third, in dirty fluids or environments with high particulate content, the small pressure ports can clog easily. In aviation, ice accumulation on the probe can lead to dangerous errors, requiring internal electrical heating elements.

- Requires velocity profile integration (e.g., Log-Tchebycheff method) to compute total discharge.
- Sensitive to misalignment; requires alignment parallel to local stream velocity.

Limitations and Troubleshooting in Operation II

- Clogging risks in particulate-heavy flows (often mitigated by gas back-purging).
- Ice buildup is resolved in flight by using integrated electrical heater coils (pitot heat).

Lecture 18: Concept of Rotameter I

Course: Fluid Mechanics and Hydraulic Machinery Topics: Variable Area Flow Meters, Working Principle of Rotameter, Derivation of Flow Rate, and Hydraulic Coefficients.

- Overview of Variable Area Flow Meters
- Detailed Analysis of Rotameter Dynamics
- Understanding Hydraulic Coefficients (Cc, Cv, Cd)

Introduction to Variable Area Flow Meters I

Unlike constant-area, variable-pressure drop meters (like orifice plates and venturi meters), a rotameter operates as a variable-area, constant-pressure drop meter. It consists of a tapered glass or plastic tube mounted vertically in the fluid pipeline, with the larger diameter at the top, and a specially designed float that is free to move vertically within the tube.

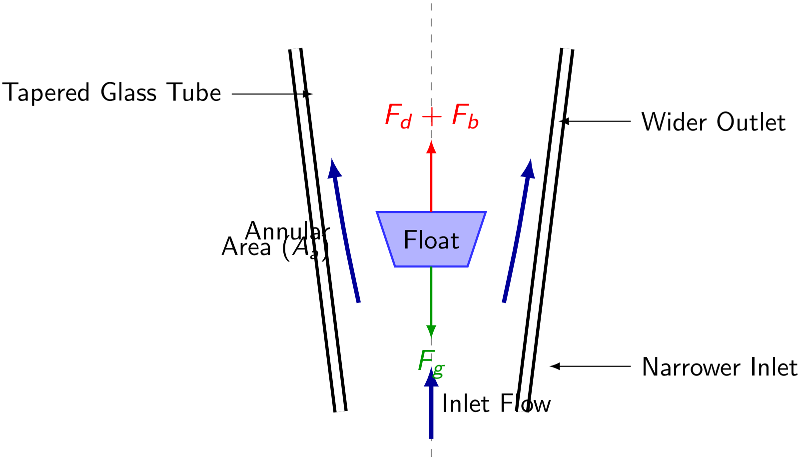
- Operates at a nearly constant pressure differential across the float.
- Fluid flow enters from the bottom of the vertically oriented tapered tube.
- The position of the float is directly proportional to the flow rate.
- Visual indication is provided by the height of the float relative to a calibrated scale.

Working Principle of Rotameter I

The operation of a rotameter is governed by a dynamic balance of forces acting on the float. When there is no flow, the float rests at the bottom of the tapered tube. When flow commences, the fluid exerts an upward drag force and buoyant force. The float rises until the upward forces exactly balance the downward gravitational force.

- Gravity acts downwards: $F_g = V_f * \rho_f * g$.
- Buoyancy acts upwards: $F_b = V_f * \rho * g$.
- Drag / pressure force acts upwards: $F_d = A_f * \Delta P$.
- At equilibrium: $F_d + F_b = F_g$, which yields $\Delta P = V_f * g * (\rho_f - \rho) / A_f$.
- Pressure drop remains constant regardless of the flow rate.

Schematic Diagram of a Rotameter I



- Tapered glass tube causes the annular area to increase as the float rises.

Schematic Diagram of a Rotameter II

- Forces acting on the float include buoyancy (Fb) and drag (Fd) upwards, and gravity (Fg) downwards.
- Annular space (Aa) between the float and the tube wall allows fluid passage.

Mathematical Derivation of Flow Rate I

To derive the volumetric flow rate (Q), we analyze the pressure drop across the float. The pressure drop (ΔP) acts across the maximum cross-sectional area of the float (A_f). Applying the force balance: $\Delta P = \frac{V_f g(\rho_f - \rho)}{A_f}$. The flow through the annular area A_a can be modeled similarly to flow through an orifice meter.

- Volumetric flow rate equation: $\mathrm{Q} = \mathrm{C_d} * \mathrm{A_a} * \sqrt{2 * g * V_\mathrm{f} * (\rho_\mathrm{f} - \rho)/(A_\mathrm{f} * \rho)}$.
- Where $\mathrm{C_d}$ is the coefficient of discharge for the rotameter.
- $\mathrm{A_a}$ is the variable annular area, which is a function of the float height (y).
- For a tube with a constant taper angle θ: $\mathrm{A_a}$ is approximately proportional to float displacement.

Relation to Float Displacement I

In a vertically tapered tube, the inner diameter D_t at any height y is given by $D_t = D_0 + 2y\tan(\theta/2)$, where D_0 is the tube diameter at the bottom and θ is the taper angle. The annular area A_a is the difference between the tube cross-sectional area and the float's maximum cross-sectional area: $A_a = \frac{\pi}{4}(D_t^2 - D_f^2)$.

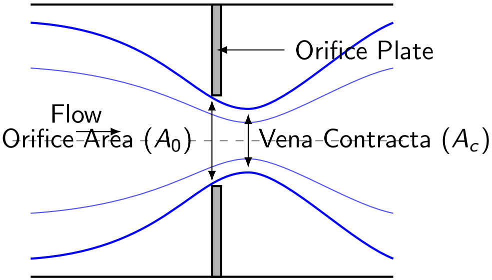
- If $\mathrm{D_0}$ is equal to the float diameter $\mathrm{D_f}$, then $\mathrm{A_a} = \pi$ * $\mathrm{D_f}$ * y * $\tan(\theta/2) + \pi$ * $\mathrm{y^2}$ * $\tan^2(\theta/2)$.
- For small taper angles, the $\mathrm{y^2}$ term is negligible, yielding a linear relationship: $\mathrm{A_a} \approx \pi$ * $\mathrm{D_f}$ * y * $\tan(\theta/2)$.
- This linear relation results in a highly desirable linear flow scale for the rotameter.
- Thus, Q proportional to y, meaning the height of the float is directly proportional to flow rate.

Introduction to Hydraulic Coefficients I

Hydraulic coefficients are dimensionless parameters used to account for the differences between theoretical and actual fluid flow behavior. In real systems, viscosity, turbulence, and flow separation create energy losses and flow contraction. The three primary coefficients are: the Coefficient of Contraction (C_c), the Coefficient of Velocity (C_v), and the Coefficient of Discharge (C_d).

- Coefficient of Contraction ($\mathrm{C_c}$): Ratio of the area of the jet at the vena contracta to the area of the orifice.
- Coefficient of Velocity ($\mathrm{C_v}$): Ratio of the actual velocity at the vena contracta to the theoretical velocity.
- Coefficient of Discharge ($\mathrm{C_d}$): Ratio of the actual discharge to the theoretical discharge.
- Fundamental relationship: $\mathrm{C_d} = \mathrm{C_c} * \mathrm{C_v}$.

Vena Contracta and Coefficients I



- The stream of fluid contracted downstream of the orifice is called the Vena Contracta.
- At the Vena Contracta, streamlines are parallel and velocity is maximum.
- C_c represents the contraction ratio: $C_c = A_c / A_0$ (typically 0.61 - 0.65 for sharp-edged orifices).
- C_v accounts for frictional losses up to the Vena Contracta (typically 0.97 - 0.99).

Hydraulic Coefficients Comparison I

While orifice plates have a low C_d (around 0.60 to 0.65) due to high contraction losses, and venturi meters have a high C_d (around 0.95 to 0.98) due to streamlined flow and minimal contraction, the rotameter's coefficient of discharge (C_d) is sensitive to float geometry and fluid viscosity. Rotameter C_d typically ranges from 0.60 to 0.80 and is characterized over a wide range of float Reynolds numbers.

- Orifice Meter: Cd is low (0.60) due to sharp contraction and sudden expansion losses.
- Venturi Meter: Cd is high (0.97) because of gradual contraction/expansion minimizing turbulence.
- Rotameter: Cd varies with float shape (e.g., plumb-bob, spherical, guided) to achieve viscosity immunity.
- Viscosity-immune floats are designed to induce turbulence deliberately, keeping Cd stable at lower Reynolds numbers.

Summary and Design Considerations I

Rotameters are highly versatile flow meters offering linear outputs, wide rangeability, and direct visual readings. However, they must be mounted vertically (flow upwards) and are sensitive to fluid density and viscosity changes, which alter the force balance on the float. Calibration is mandatory for specific fluid properties to ensure accurate readings in industrial applications.

- Rotameters are constant pressure drop, variable area flow meters.
- The height of the float is linear with flow rate under small tube taper angles.
- Hydraulic coefficients (Cc, Cv, Cd) describe performance deviations from the ideal model.
- Selection of float shape and density depends on fluid compatibility and desired Reynolds range.

Major and Minor (concept) losses I

Course: Fluid Mechanics and Hydraulic Machinery

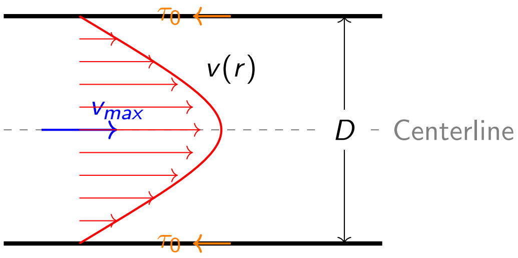
- Lecture 19: Fluid transport energy dissipation
- Continuous viscous friction vs. localized turbulence losses
- Analytical derivation and design application

Topic Details I

Energy degradation in piping systems is a primary consideration in hydraulic engineering. As a fluid flows through a conduit, viscosity and boundary shear stress convert mechanical energy into thermal energy. This loss of total head is categorized into major frictional loss, which is distributed continuously over the entire length of the pipe, and minor losses, which occur at local obstructions such as elbows, bends, valves, expansions, and contractions. For proper system design, pumps must be sized to overcome the cumulative head losses, ensuring that the desired flow rate is delivered to the system outlet.

- Head loss represents energy loss per unit weight of fluid.
- Major losses are proportional to pipe length, whereas minor losses occur locally.
- The ratio of minor to major losses determines whether minor losses can be neglected.

Velocity Profile and Shear Stress I

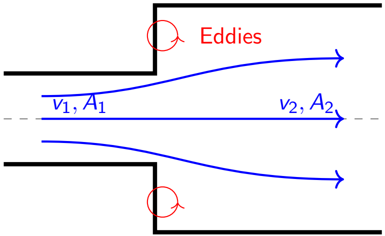


- Boundary layer development leads to shear stress at the pipe wall.
- Velocity is zero at the solid boundary due to the no-slip condition.
- Frictional loss is directly related to the wall shear stress.

The major head loss in a circular pipe is mathematically modeled by the Darcy-Weisbach equation: $h_f = \frac{fLv^2}{2gD}$, where f is the Darcy friction factor. In open-channel flow and large conduits, the Chezy equation is often applied: $v = C\sqrt{mi}$, where C is the Chezy discharge coefficient, m is the hydraulic mean depth (A/P), and i is the hydraulic slope (h_f/L). For a circular pipe running full, the hydraulic mean depth $m = D/4$, establishing the functional relationship between the friction factor and Chezy coefficient as $C = \sqrt{8g/f}$.

- For laminar flow ($Re < 2000$), the friction factor is $f = 64/Re$ (independent of roughness).
- For turbulent flow ($Re > 4000$), f is determined by the Colebrook-White equation.
- Roughness ratio (ϵ/D) plays a critical role in turbulent frictional resistance.

Sudden Pipe Expansion Flow Field I



- Flow separation occurs at the sharp corners of the boundary transition.
- Turbulent eddies form in the corners, dissipating mechanical energy.
- Head loss is proportional to the square of the difference in velocities: $h_e = (v_1 - v_2)^2/2g$.

Minor Losses: Analytical Expressions I

Minor losses (h_m) are modeled using empirical loss coefficients (K) applied to the velocity head: $h_m = K\frac{v^2}{2g}$. For a sudden contraction, the head loss is $h_c = (\frac{1}{C_c} - 1)^2 \frac{v_2^2}{2g} \approx 0.5\frac{v_2^2}{2g}$, where C_c is the coefficient of contraction. The entrance loss for a sharp-edged pipe inlet is $h_{in} = 0.5\frac{v^2}{2g}$, while a rounded entrance reduces K to 0.04. Pipe exit losses correspond to $h_{out} = 1.0\frac{v^2}{2g}$ as the kinetic energy of the fluid is fully dissipated in the receiving reservoir.

- Valves and fittings introduce minor losses based on their internal flow path geometries.
- A globe valve ($K \approx 10$) creates significantly more head loss than a gate valve ($K \approx 0.15$).
- Bends and elbows create secondary flows, generating loss coefficients that vary with radius of curvature.

Hydraulic Gradient and Total Energy Lines I

Analyzing pipelines requires constructing the Total Energy Line (TEL) and the Hydraulic Gradient Line (HGL). The TEL represents the total energy head: $TEL = \frac{p}{\rho g} + z + \frac{v^2}{2g}$, and slopes continuously downward in the direction of flow due to frictional head loss (h_f). The HGL represents the piezometric head: $HGL = \frac{p}{\rho g} + z$. The vertical distance between the two lines is the dynamic velocity head, $\frac{v^2}{2g}$. Sudden contractions or expansions produce rapid changes in the HGL profile, while valves and bends create distinct downward steps in the TEL.

- TEL must always decrease along the flow path unless energy is added by a pump.
- If the HGL drops below the physical pipeline elevation, the pressure is sub-atmospheric (risk of cavitation).
- The slope of the HGL is called the hydraulic gradient.

Numerical Problem 1: Darcy vs. Chezy I

Problem: Water flows through a horizontal pipe of diameter 300 mm and length 1500 m at a velocity of 2.5 m/s. Calculate the head loss due to friction using (a) Darcy-Weisbach formula with $f = 0.02$, and (b) Chezy's formula with $C = 60$.

Solution: (a) Darcy-Weisbach:
$h_f = \frac{fLv^2}{2gD} = \frac{0.02 \times 1500 \times 2.5^2}{2 \times 9.81 \times 0.3} = 31.86$ m. (b) Chezy:
$v = C\sqrt{mi} \Rightarrow 2.5 = 60\sqrt{(0.3/4) \cdot i}$. Solving for i, we find
$i = 0.02315$. Since $i = h_f/L$, then $h_f = 0.02315 \times 1500 = 34.72$ m.

- Input parameters must be converted to SI units prior to calculation.
- Hydraulic mean depth (m) is $D/4$ for a full-flowing circular pipe.
- The variation in results demonstrates that Chezy's C and Darcy's f make different baseline assumptions.

Numerical Problem 2: Combined Losses I

Problem: A pipeline of 200 mm diameter and 100 m length connects two reservoirs. The difference in water levels is 15 m. The pipe has a sharp-edged entrance, a fully open gate valve ($K = 0.2$), two 90° bends ($K = 0.5$ each), and a sudden outlet. Calculate the flow rate Q if $f = 0.03$.

Solution: Apply energy equation between reservoir surfaces:
$H = \sum h_{losses} = (K_{in} + \frac{fL}{D} + K_{valve} + 2K_{bend} + K_{out})\frac{v^2}{2g}$
$15 = (0.5 + \frac{0.03 \times 100}{0.2} + 0.2 + 2(0.5) + 1.0)\frac{v^2}{2g} = 17.7\frac{v^2}{2g}$ Velocity
$v = \sqrt{\frac{15 \times 2 \times 9.81}{17.7}} = 4.08$ m/s. Discharge
$Q = Av = \frac{\pi}{4}(0.2)^2 \times 4.08 = 0.128$ m^3/s.

- In this system, major friction loss ($15v^2/2g$) is the dominant component of total loss.
- Minor losses contribute approximately 15

Numerical Problem 2: Combined Losses II

- Discharge is the volumetric flow rate computed as velocity times cross-sectional area.

Summary and Engineering Recommendations I

Evaluating major and minor losses is fundamental for pipeline network optimization and hydraulic machinery integration. Major losses dominate in long-distance transportation lines, while minor losses become critical in plant piping networks with dense distributions of valves, fittings, and bends. Mitigating minor losses through design changes—such as substituting standard elbows with long-radius bends and minimizing sudden diameter changes—reduces total system resistance. This directly lowers pump size requirements and operating energy consumption.

- Always sum both major and minor losses to size booster pumps correctly.
- Use gradual expansions (diffusers) to recover kinetic energy and reduce minor head loss.
- Aging pipelines exhibit increased roughness, leading to higher friction factors and increased power requirements over time.

Reynolds' Experiment I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 20: Reynolds' Experiment & Flow Regimes Department of Mechanical Engineering

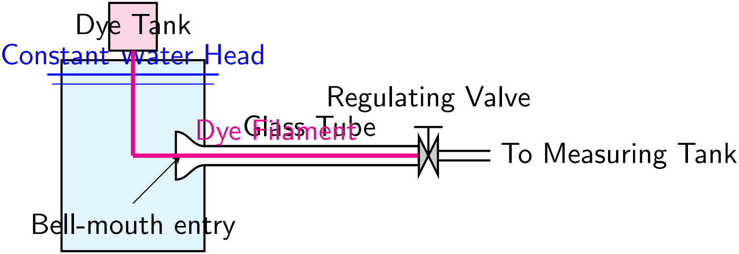
- Introduction to laminar and turbulent flows.
- Osborne Reynolds' seminal experimental study (1883).
- Significance of critical velocities in engineering pipe design.

Experimental Objectives I

The primary objective of Reynolds' experiment is to study the characteristics of flow of water through a glass tube. By varying the velocity of flow, the experiment demonstrates the transition from laminar to turbulent flow, identifies critical velocities, and establishes the relationship between flow regimes and a dimensionless parameter known as the Reynolds Number.

- Visualize laminar, transitional, and turbulent flow regimes.
- Determine the lower and upper critical velocities of flow.
- Understand the physical factors influencing flow instability.
- Demonstrate the concept of dynamic similarity using the Reynolds Number.

Reynolds' Experimental Setup I

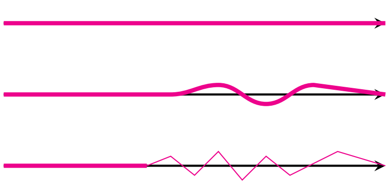


- Constant-head water tank maintains a uniform pressure gradient during flow.
- Bell-mouth entry prevents flow separation and boundary layer disturbance.
- Dye container supplies a fine, colored filament at the center of the pipe inlet.
- Regulating valve adjusts discharge to control flow velocity dynamically.

The experiment is conducted by filling the tank with water and allowing it to stand until all disturbances die out. The regulating valve is opened slowly to establish a very low velocity. Dye is then injected into the water stream, and its behavior is observed. The flow rate is measured using a stopwatch and a calibrated measuring tank. The valve opening is gradually increased to observe the transition, and then closed slowly to observe the reverse transition.

- Allow water to settle completely to ensure zero initial turbulent energy.
- Inject dye at a rate matching the surrounding water velocity (isokinetic injection).
- Record volume flow rate (Q) and time (t) to calculate average velocity ($V = Q/(A \cdot t)$).
- Repeat for both increasing flow rate (loading) and decreasing flow rate (unloading).

Visualizing the Three Flow Regimes I



Laminar (Re ¡ 2000)

Transitional (2000 ¡ Re ¡ 4000)

Turbulent (Re ¿ 4000)

- Laminar flow: The dye remains as a thin, stable, straight line along the tube axis.
- Transitional flow: The dye filament begins to oscillate and show wavy instabilities.
- Turbulent flow: The dye filament breaks up, diffuses rapidly, and colors the entire fluid.

Laminar vs. Turbulent Flow Dynamics I

In laminar flow, fluid particles move in well-defined paths or streamlines, sliding over adjacent layers without macroscopic mixing. Shear stresses are governed purely by Newton's law of viscosity. In contrast, turbulent flow is characterized by the random, chaotic motion of fluid particles. Eddy currents and transverse velocity components cause intense mixing, which leads to significantly higher energy dissipation and pressure drop.

- Laminar shear stress: $\tau = \mu \frac{du}{dy}$ is linear and depends on viscosity.
- Turbulent shear stress: $\tau = (\mu + \eta) \frac{du}{dy}$ where η is eddy viscosity.
- Velocity profile: Laminar is parabolic ($u_{max}/V_{avg} = 2$), while turbulent is flatter/logarithmic ($u_{max}/V_{avg} \approx 1.2$).

Laminar vs. Turbulent Flow Dynamics II

- Energy loss: Head loss is proportional to velocity ($h_f \propto V$) in laminar flow, and approximately to velocity squared ($h_f \propto V^2$) in turbulent flow.

Formulation of the Reynolds Number I

Osborne Reynolds deduced that the transition from laminar to turbulent flow is governed by a dimensionless ratio representing the relative importance of inertial forces to viscous forces within the fluid. This ratio is defined as the Reynolds Number (Re). For flow in a circular pipe of diameter D, velocity V, density ρ, and dynamic viscosity μ, the number is calculated as $Re = \frac{\rho VD}{\mu} = \frac{VD}{\nu}$ where ν is the kinematic viscosity.

- Inertial force: Inertial Force $\propto \rho V^2 D^2$ (mass flow rate times velocity change).
- Viscous force: Viscous Force $\propto \mu VD$ (shear stress times area).
- Dimensionless parameter: Re has no units and is identical in any coherent unit system.
- Physical meaning: High Re indicates inertial forces dominate; low Re indicates viscous forces damp out perturbations.

Lower and Upper Critical Reynolds Numbers I

The transition from laminar to turbulent flow does not occur at a single velocity but over a range. The lower critical Reynolds number ($Re_{crit,lower}$) is the value below which any flow disturbance is damped out by viscous forces, ensuring the flow is always laminar. The upper critical Reynolds number ($Re_{crit,upper}$) is the value above which any minor disturbance triggers transition to turbulence. Under normal engineering conditions, these thresholds define the flow regimes in circular pipes.

- Lower Critical Reynolds Number: $Re \approx 2000$ (practically, flow remains laminar).
- Upper Critical Reynolds Number: $Re \approx 4000$ (flow is fully turbulent).
- Transitional Range: $2000 < Re < 4000$ (unstable flow alternating between regimes).

Lower and Upper Critical Reynolds Numbers II

- With extreme care and vibration-free environments, laminar flow can be maintained up to $Re \approx 100,000$.

Friction Factor and Head Loss Relations I

The Reynolds Number directly determines the friction factor (f) used in the Darcy-Weisbach equation for head loss ($h_f = f \frac{L}{D} \frac{V^2}{2g}$). In the laminar regime, the friction factor is independent of pipe roughness and is a simple function of Re. In the turbulent regime, the friction factor depends on both the Reynolds number and the relative roughness of the pipe wall (as illustrated in the Moody diagram).

- Laminar friction factor: $f = \frac{64}{Re}$ (derived from Hagen-Poiseuille law).
- Turbulent friction factor: Colebrook-White equation or Haaland approximation is used.
- At very high Re, the flow becomes 'fully rough' and f becomes independent of Re.
- Accurate prediction of Re is essential to size pumps, compressors, and pipeline networks.

Summary & Practical Engineering Insights I

Reynolds' experiment remains one of the most fundamental studies in fluid mechanics. It demonstrated the transition from laminar to turbulent flow and introduced the concept of dimensionless parameters for scaling and similarity. In practice, engineers must identify the flow regime to accurately estimate pressure drop, heat transfer coefficients, and mixing rates in industrial equipment.

- Laminar flow is preferred in lubrication systems and microfluidics to minimize mixing and shear.
- Turbulent flow is essential in heat exchangers and chemical reactors to maximize heat and mass transfer.
- Pipe network design utilizes Re to choose appropriate friction models and minimize pumping power.
- Dynamic similarity: Scale models in wind tunnels and towing tanks must match Re for valid hydrodynamic testing.

Course: Fluid Mechanics and Hydraulic Machinery Lecture 21: Derivations, Physical Insights, and Engineering Calculations of Frictional Head Loss in Pipes and Channels

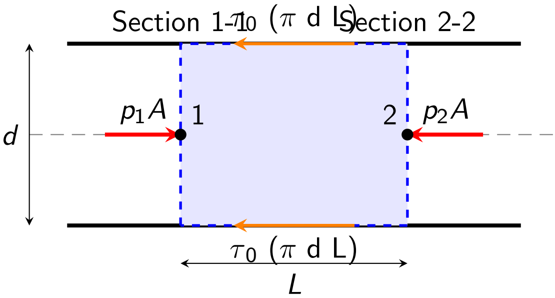
- Introduction to head loss in fluid flow systems
- Derivation of the Darcy-Weisbach equation from first principles
- Derivation of Chezy's equation for open channel and pipe flow
- Analysis of friction factors and empirical coefficients
- Practical step-by-step engineering numerical problems

Frictional Head Loss in Pipe Flow I

When a real fluid flows through a pipe, it experiences resistance due to viscosity and boundary shear stress. This resistance converts useful mechanical energy into thermal energy, resulting in a continuous loss of pressure head along the direction of flow. Understanding and quantifying this head loss is critical for designing pipelines, pumps, and water distribution networks.

- Frictional loss is the primary component of major losses in long pipelines.
- The loss is directly related to the shear stress exerted by the fluid on the pipe wall.
- Flow regimes (laminar vs. turbulent) determine the behavior of boundary resistance.
- Darcy-Weisbach and Chezy equations are the primary analytical tools used to quantify friction loss.

Force Balance on a Fluid Control Volume I



- A cylindrical control volume of length L and diameter d is analyzed for steady, uniform flow.
- Forces acting on the fluid element include pressure forces (p1*A and p2*A) and the shear force (t0*pi*d*L).
- Under steady state, the sum of forces in the direction of flow must be equal to zero.

Force Balance on a Fluid Control Volume II

- This force balance forms the foundation for deriving both Darcy's and Chezy's equations.

Derivation of the Darcy-Weisbach Equation I

Let's perform a force balance on a horizontal pipe of diameter d and length L: $(p_1 - p_2)A - \tau_0(\pi dL) = 0$ Dividing by the specific weight $\gamma = \rho$ g, we express the pressure drop as head loss h_f: $h_f = \frac{p_1 - p_2}{\gamma} = \frac{4\tau_0 L}{\gamma d}$ To relate this to flow velocity, we introduce Froude's definition of shear stress in turbulent flow: $\tau_0 =$ f' $\frac{\rho v^2}{2}$, where f' is the coefficient of friction. Substituting this yields: $h_f = \frac{4f' L v^2}{2gd}$. Defining the Darcy friction factor f = 4f' gives the final Darcy-Weisbach equation: $h_f = \frac{fLv^2}{2gd}$

- Head loss is directly proportional to length L and velocity squared v^2, and inversely proportional to diameter d.
- The Darcy friction factor f is a dimensionless quantity that characterizes pipe roughness and Reynolds number.
- For laminar flow (Re ¡ 2000), f = 64/Re, which can be derived analytically from Hagen-Poiseuille flow.

Derivation of the Darcy-Weisbach Equation II

- For turbulent flow, f is determined using the Moody chart or Colebrook-White equation.

Darcy Friction Factor & Flow Regimes I

The Darcy friction factor (f) is not a constant; its value depends heavily on the Reynolds number (Re) and the relative roughness of the pipe wall (e/d). In laminar flow, the friction factor is independent of pipe roughness and is given by $f = 64/Re$. In fully turbulent flow, the laminar sublayer is broken, and the friction factor becomes independent of the Reynolds number, relying solely on relative roughness.

- Laminar Flow: f = 64/Re (independent of pipe material or roughness).
- Transition Zone (2000 ¡ Re ¡ 4000): Flow behavior is highly unstable and difficult to predict.
- Turbulent Flow (Smooth Pipes): f is a function of Re alone (Blasius equation: f = 0.3164 / Re^0.25 for Re ¡ 10^5).

Darcy Friction Factor & Flow Regimes II

- Fully Rough Turbulent Flow: f depends only on relative roughness (e/d), which is represented by horizontal lines on the Moody Chart.

Derivation of the Chezy Equation I

The Chezy equation is widely used for open channel flow and large conduits. Starting from the force balance: $(p_1 - p_2)A = \tau_0 PL$, where P is the wetted perimeter ($P = \pi d$ for full pipe). Writing $\frac{p_1 - p_2}{\gamma} = \mathrm{h_f}$, we get $\tau_0 = \gamma \ \frac{A}{P} \ \frac{h_f}{L}$. We define the hydraulic radius m $=$ A/P (often denoted as R) and the hydraulic slope i $= \mathrm{h_f}/\mathrm{L}$. Thus: $\tau_0 = \gamma$ m i. Substituting the experimental relation $\tau_0 = \mathrm{k\ v^2}$ (where k is a constant), we get: $\mathrm{k\ v^2} = \rho$ g m i $\Rightarrow$ v $= \sqrt{\frac{\rho g}{k}} \ \sqrt{mi}$.

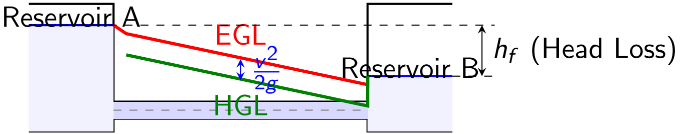
Defining Chezy's constant C $= \sqrt{\frac{\rho g}{k}}$, we obtain: v $=$ C $\sqrt{mi}$

- Chezy's equation expresses velocity as a function of hydraulic radius m, hydraulic slope i, and Chezy coefficient C.
- Hydraulic radius (m) represents the ratio of flow cross-sectional area to wetted perimeter.
- Hydraulic slope (i) represents the head loss per unit length of the channel or pipe.

Derivation of the Chezy Equation II

- Chezy's coefficient C has dimensions of $L^{(1/2)}/T$ ($m^{(1/2)}/s$) and is not dimensionless like Darcy's f.

Hydraulic and Energy Gradient Lines I



- Energy Gradient Line (EGL) represents the total head (pressure head + datum head + velocity head) along the pipe.
- Hydraulic Gradient Line (HGL) represents the piezometric head (pressure head + datum head) along the pipe.
- The vertical distance between EGL and HGL is equal to the dynamic head ($v^2/2g$).
- Head loss due to friction is shown by the downward slope of both EGL and HGL.

Relationship between Darcy's f and Chezy's C I

Although Darcy's and Chezy's equations were developed through different approaches, they both quantify major head loss in pipes and are mathematically related. From Darcy-Weisbach: $\mathrm{h_f} = \frac{fLv^2}{2gd}$ $\Rightarrow \mathrm{v}^2 = \frac{2gdh_\mathrm{f}}{fL}$. For a circular pipe, the hydraulic radius is: $\mathrm{m} = \frac{A}{P}$ $= \frac{\pi d^2/4}{\pi d} = \frac{d}{4} \Rightarrow \mathrm{d} = 4\mathrm{m}$. Substituting $\mathrm{d} = 4\mathrm{m}$ and hydraulic slope $\mathrm{i} = \mathrm{h_f}/\mathrm{L}$: $\mathrm{v}^2 = \frac{2g(4m)i}{f} = \frac{8g}{f}\ \mathrm{m\ i} \Rightarrow \mathrm{v} = \sqrt{\frac{8g}{f}}\sqrt{mi}$. Comparing this with Chezy's formula ($\mathrm{v} = \mathrm{C}\sqrt{mi}$), we establish: $\mathrm{C} = \sqrt{\frac{8g}{f}}$

- Chezy's C is inversely proportional to the square root of the Darcy friction factor f.
- Since g is constant, C varies directly with the smoothness of the pipe (higher C means lower friction factor f).
- This relation allows engineers to convert between Chezy's and Darcy's equations when analyzing pipe flows.

- Unlike f, which is dimensionless, Chezy's C has dimensions of $L^{(1/2)}/T$ (e.g., $m^{(1/2)}/s$).

Numerical Problem 1: Darcy's Equation I

Problem: Water flows through a horizontal pipe of diameter 300 mm and length 1.5 km at a velocity of 2 m/s. Calculate the head loss due to friction using Darcy's equation if the coefficient of friction f' = 0.005.
Solution Steps: 1. Given: Diameter d = 0.3 m, Length L = 1500 m, Velocity v = 2 m/s. 2. The friction coefficient f' = 0.005. Note that Darcy friction factor f = 4f' = 4 × 0.005 = 0.02. 3. Apply Darcy-Weisbach equation: $h_f = \frac{fLv^2}{2gd}$ 4. Substitute values: $h_f = \frac{0.02 \times 1500 \times 2^2}{2 \times 9.81 \times 0.3}$ 5. Calculate: $h_f = \frac{120}{5.886} = 20.39$ m of water.

- Ensure whether the problem specifies friction factor (f) or friction coefficient (f'). Here f = 4 * f'.
- Check that all dimensions are converted to standard SI units (meters, seconds) before calculation.

Numerical Problem 1: Darcy's Equation II

- The head loss of 20.39 m represents a significant pressure loss that would require pump energy to overcome.
- Velocity has a quadratic effect on head loss; doubling velocity to 4 m/s increases loss to 81.56 m.

Numerical Problem 2: Chezy's Equation I

Problem: Find the velocity of flow and discharge through a circular pipe of diameter 1.2 m laid at a slope of 1 in 1000. Take Chezy's constant C = 55. The pipe is running half full.
Solution Steps: 1. Given: Diameter d = 1.2 m, Bed Slope i = 1/1000 = 0.001, C = 55. 2. For a pipe running half full: Area A = $\frac{1}{2}$ $(\frac{\pi d^2}{4})$ = 0.5655 m^2. Wetted perimeter P = $\frac{1}{2}$ (π d) = 1.885 m. 3. Hydraulic radius: m = A/P = d/4 = 1.2 / 4 = 0.3 m. 4. Velocity: v = C $\sqrt{mi}$ = 55 $\times$ $\sqrt{0.3 \times 0.001}$ = 55 $\times$ 0.01732 = 0.9526 m/s. 5. Discharge: Q = A $\times$ v = 0.5655 $\times$ 0.9526 = 0.5387 m^3/s.

- For both half-full and full pipes, the hydraulic radius m equals d/4 (since area and perimeter are halved).
- Chezy's C incorporates the effect of boundary roughness, and a higher value increases velocity.

Numerical Problem 2: Chezy's Equation II

- Discharge (Q) represents the volumetric flow rate, calculated by multiplying cross-sectional area by average velocity.
- Slope i (1 in 1000) drives the gravity flow, and a steeper slope would increase both velocity and discharge.

Water hammer its cause I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 22: Analysis of Water Hammer Phenomenon, Its Causes, Governing Equations, and Physical Behavior in Piping Systems.

- Focus: Transient Flow in Closed Conduits
- Prerequisites: Fluid Properties, Continuity, Momentum, and Wave Theory

Introduction to Water Hammer I

Water hammer, or hydraulic shock, is a transient pressure surge that occurs when a moving fluid is suddenly forced to stop or change direction, typically by the rapid closure of a valve. This sudden change in momentum generates high-pressure waves that propagate along the pipeline, potentially causing pipe rupture, joint failure, or damage to pumps and turbines.

- Occurs in closed conduits running full under pressure.
- Driven by conversion of kinetic energy into elastic strain energy.
- Characterized by high-pressure waves traveling at the sonic velocity of the fluid medium.
- Requires analyzing fluid compressibility and pipe wall elasticity.

Physical Cause and Mechanism I

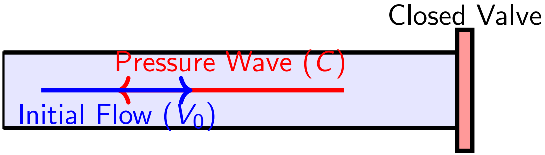


Figure: Propagation of pressure wave upstream upon sudden valve closure.

- Downstream valve closes, bringing adjacent fluid particles to rest.
- Kinetic energy is converted to pressure head and elastic strain energy.
- A compression wave propagates upstream at sonic velocity C.
- Fluid behind the wave front is compressed and has zero velocity.

The Speed of the Pressure Wave (Sonic Velocity) I

The speed C of the pressure wave propagation depends on fluid bulk modulus K, density ρ, pipe material Young's modulus E, diameter D, and wall thickness e. For rigid pipes, the wave speed is $C_0 = \sqrt{K/\rho}$. Pipe elasticity reduces the effective bulk modulus, decreasing the wave propagation speed.

- Wave speed equation: $C = \sqrt{\frac{K/\rho}{1+(K/E)(D/e)}}$.
- Wave speed in steel water pipes ranges from 1000 to 1200 m/s.
- Thicker walls and stiffer materials yield higher wave speeds.
- Entrained air or gas bubbles drastically reduce K and lower C.

Joukowsky Equation for Sudden Closure I

The fundamental equation for maximum pressure rise due to sudden valve closure was derived by Joukowsky in 1898. For rapid closure (where the closure time is less than the wave's round-trip travel time), the pressure rise ΔP is directly proportional to fluid density, wave speed, and change in fluid velocity.

- Joukowsky Equation: $\Delta P = \rho C \Delta V$.
- Pressure head rise format: $\Delta H = \frac{C \Delta V}{g}$.
- A velocity change of 1 m/s in steel pipes causes a pressure surge of 10 bar (100 m head).
- Peak pressure is independent of pipeline length for sudden closures.

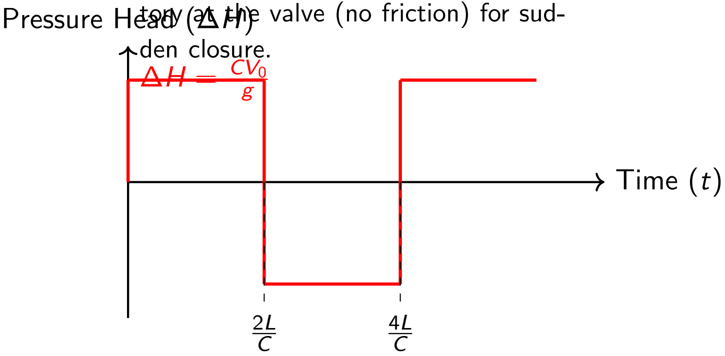
Classification of Valve Closure Time I

Valve closure is classified as rapid (sudden) or gradual (slow) based on the critical time of wave reflection, $T_r = 2L/C$. If the valve is closed completely before the pressure wave travels to the reservoir and returns to the valve as a relief wave, the closure is rapid.

- Critical Reflection Time: $T_r = \frac{2L}{C}$.
- Rapid Closure: $T_c \leq \frac{2L}{C}$ (produces maximum pressure surge $\Delta H = \frac{CV_0}{g}$).
- Gradual Closure: $T_c > \frac{2L}{C}$ (reflected relief waves reach the valve before it closes completely).
- Gradual closure peak pressure rise (Allievi formula): $\Delta H \approx \frac{2LV_0}{gT_c}$.

Pressure Wave Propagation Phases I

Figure: Theoretical pressure head history at the valve (no friction) for sudden closure.



- Phase 1 ($0 < t < 2L/C$): Valve pressure remains high at $+\Delta H$ as the wave travels.
- Phase 2 ($2L/C < t < 4L/C$): Pressure drops to $-\Delta H$ (suction wave) due to fluid flow reversal.

Pressure Wave Propagation Phases II

- Phase 3 ($t > 4L/C$): The cycle repeats, oscillating between positive and negative heads.
- Hydraulic friction gradually dampens the pressure waves in real pipelines.

Column Separation and Cavitation I

During the negative pressure phase ($2L/C < t < 4L/C$), if the absolute pressure drops below the vapor pressure of the liquid, the water vaporizes and forms cavities. This is called column separation. When the flow reverses and pressure rises, the vapor cavity collapses violently, creating massive localized pressure spikes.

- Vapor pressure of water at $20^{\circ}C$ is ≈ 2.34 kPa absolute.
- Cavity collapse can produce pressure shocks exceeding the initial Joukowsky limit.
- Commonly occurs at high elevation points along the pipeline.
- Mitigated using vacuum breaker valves and air release valves.

Factors Influencing Transient Severity I

The magnitude of water hammer transient pressures is determined by various design parameters of the pipeline system. Engineers must evaluate these factors during the layout design stage to ensure the pipe thickness and valve specifications can withstand the transients.

- Flow Velocity (V_0): Proportional relationship; minimizing design velocity reduces surge risks.
- Pipeline Length (L): Longer lines lengthen the reflection time, increasing the risk of rapid closure.
- Material Elasticity (E): Materials with lower modulus (e.g., HDPE) have lower wave speeds and surges.
- Valve Closure Curve: Linear vs. non-linear closure; butterfly and gate valves require special care.

Mitigation and Control Devices I

Engineers install active and passive devices to prevent water hammer. The choice of mitigation device depends on pipeline dimensions, topography, pump characteristics, and cost constraints.

- Slow-Closing Valves: Extending closure time T_c to be significantly greater than $2L/C$.
- Surge Tanks: Open-topped reservoirs to store excess fluid and convert kinetic energy to potential energy.
- Air Vessels: Pressurized accumulators with gas volumes that absorb pressure surges.
- Pressure Relief Valves: Safety valves that open to discharge water when pressure thresholds are exceeded.

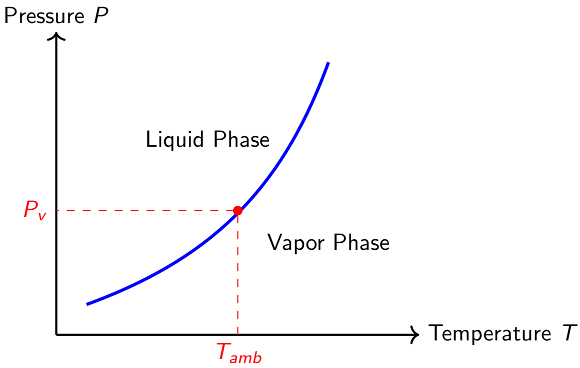
Course: Fluid Mechanics and Hydraulic Machinery Lecture 23: Cavitation Effects and Remedies in Hydraulic Machinery

- Course: Fluid Mechanics and Hydraulic Machinery
- Instructor: Professor of Mechanical Engineering
- Topic: Cavitation Phenomena, Effects, and Remedies

Understanding Cavitation I

Cavitation occurs when the local pressure in a liquid stream falls below its vapor pressure at the prevailing temperature. This leads to rapid boiling and formation of vapor bubbles, which subsequently collapse with extreme violence when carried into high-pressure regions.

- Mainly occurs in low-pressure zones of pumps (suction side of impeller) and turbines (draft tube entrance/runner exit).
- Vapor bubbles form because liquid molecules separate due to tension rather than thermal energy.
- The subsequent collapse of these bubbles is the primary driver of cavitation damage.



Cavitation occurs when pressure drops below P_v at T_{amb}.

- The vapor pressure curve represents the phase boundary where boiling occurs.

Vapor Pressure and Phase Transitions II

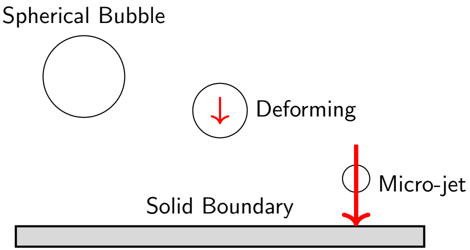
- Isothermal depressurization occurs when fluid velocity increases drastically (Bernoulli's Principle).
- If pressure falls below P_v, vapor cavities form instantly.

Dynamics of Bubble Collapse I

When vapor bubbles enter high-pressure areas, they collapse in microseconds. Due to the proximity of solid walls, the collapse is highly asymmetric, producing high-velocity micro-jets (up to 1000 m/s) directed toward the boundary surface.

- Bubble collapse releases energy within extremely small spatial scales.
- Micro-jets puncture the vapor pocket, striking the solid surface with high kinetic energy.
- High localized pressure pulses (up to 10 GPa) are generated at the boundary.

Asymmetric Bubble Collapse Diagram I



Stages of bubble collapse near a solid surface.

- First Stage: Symmetric bubble in a uniform pressure field.
- Second Stage: Asymmetric deformation due to flow boundary proximity.
- Third Stage: Collapse and formation of high-velocity micro-jet hitting the surface.

Effects of Cavitation on Hydraulic Machinery I

Cavitation has severe consequences on the mechanical integrity and performance of hydraulic machines, including physical damage, hydraulic loss, and structural vibrations.

- Material Erosion: Repeated pressure shocks cause fatigue failure, leading to pitting and sponge-like cavities.
- Vibration and Noise: Collapse events generate high-frequency pressure waves, causing noise and potential structural fatigue.
- Performance Degradation: Vapor pockets reduce the active flow area, causing drops in head, discharge, and efficiency.

Thoma's Cavitation Factor (σ) I

Thoma's cavitation factor is a key dimensionless parameter used in turbine design to evaluate the likelihood of cavitation. It compares the suction head margin to the total head across the machine.

- Defined as: $\sigma = \frac{H_b - H_s - h_f}{H}$, where H_b is barometric head, H_s is suction head, and H is total head.
- The critical cavitation factor σ_c marks the limit where cavitation begins to occur.
- For safe operation, the operating cavitation factor σ must be kept strictly above σ_c.

Net Positive Suction Head (NPSH) I

In rotodynamic pumps, Net Positive Suction Head represents the difference between suction pressure and vapor pressure. Maintaining a positive NPSH margin is critical to ensure fluid remains in the liquid state.

- $NPSH_{avail} = \frac{P_{atm}}{\rho g} - \frac{P_v}{\rho g} - h_s - h_{fs}$.
- $NPSH_{req}$ is a pump characteristic determined by the manufacturer.
- To avoid cavitation, the safety margin $NPSH_{avail} > NPSH_{req}$ must always be satisfied.

Remedies and Mitigation: Design & Materials I

Engineers employ robust design principles and select specific materials to mitigate cavitation damages and prevent low-pressure zones.

- Installation Height: Lowering the elevation of the machine relative to the tailrace reduces the suction lift (H_s).
- Cavitation-Resistant Materials: Using stainless steel (18-8 Cr-Ni) or alloy steels increases fatigue limits.
- Protective Coatings: Applying cobalt-base alloys (Stellite) or epoxy coatings provides sacrificial protection.

Remedies and Mitigation: Operations I

Operational parameters can be adjusted dynamically or through maintenance routines to minimize cavitation during off-design conditions.

- Air Injection: Introducing a controlled amount of air into the draft tube or low-pressure zone cushions the bubble collapse.
- Surface Polishing: Keeping runner blades highly polished reduces localized separation points and boundary layer turbulence.
- Avoiding Off-Design Operation: Preventing prolonged operation at low loads or part loads where vortex ropes form.

Course: Fluid Mechanics and Hydraulic Machinery Instructor: Professor of Mechanical Engineering

Introduction to Jet Impact & Impulse-Momentum Principle I

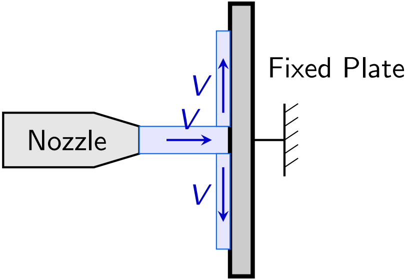
The study of jet impact forms the physical foundation for the analysis and design of impulse turbomachinery, such as Pelton water wheels and steam turbine stages. The hydrodynamic forces exerted by fluid jets on stationary or moving boundaries are governed by Newton's Second Law of Motion and the Impulse-Momentum Principle. When a fluid jet with high kinetic energy strikes a solid boundary, its velocity vector changes, resulting in a rate of change of momentum. The solid surface exerts an equal and opposite force on the fluid to cause this flow deflection, which manifests as a dynamic thrust force on the plate.

- Governed by the impulse-momentum equation: $F = d(m*V)/dt = \rho * Q * \Delta V$.
- High-velocity jet is produced by converting fluid pressure head into kinetic energy using a convergent nozzle.

Introduction to Jet Impact & Impulse-Momentum Principle II

- Understanding forces on stationary (fixed) boundaries is the first step before modeling moving boundaries where mechanical work is done.

Diagram: Normal Jet Striking a Fixed Vertical Flat Plate I



- A horizontal jet of diameter 'd' and velocity 'V' strikes the flat plate at 90 degrees.
- The plate is rigid and fixed, causing the jet to split symmetrically into two vertical streams.
- The fluid exits parallel to the plate face, reducing the normal velocity component to zero.

Force Analysis: Fixed Vertical Flat Plate I

To determine the hydrodynamic force exerted by the jet on a fixed vertical flat plate, we apply the linear momentum equation along a coordinate axis normal to the plate (the x-axis). The system is assumed to be steady, frictionless, and open to the atmosphere. Since the plate is vertical, the horizontal component of velocity after deflection is zero ($V_{2x} = 0$). The rate of momentum entering the control volume is compared to the rate of momentum leaving it, yielding the dynamic force on the plate.

- Mass flow rate of the jet: $m_{dot} = \rho * A * V$, where A is cross-sectional area and ρ is fluid density.
- Inlet momentum flux in x-direction: $M_{in} = m_{dot} * V = \rho * A * V^2$.
- Outlet momentum flux in x-direction: $M_{out} = m_{dot} * V_{2x} = 0$.
- Normal force exerted on the plate: $F_x = M_{in} - M_{out} = \rho * A * V^2$.

Energy Considerations and Boundary Layer Effects I

In an ideal fluid flow, the total mechanical energy is conserved along the streamlines according to Bernoulli's equation. Since pressure at the inlet and outlet of the control volume is atmospheric, the velocity of flow along the surface of the plate remains equal to the initial jet velocity V. However, in real fluids, boundary layer growth occurs as the fluid spreads radially over the plate. This creates wall shear stress, reducing the velocity of the fluid leaving the plate boundaries and slightly decreasing the reaction forces.

- Stagnation point exists at the center of impact where velocity is zero and pressure is maximum.
- Stagnation pressure is $P_{stag} = 0.5 * \rho * V^2$, which represents the dynamic head.
- Viscous drag forces reduce the velocity at the plate boundary: $V_{exit} = k * V$, where k ¡ 1.

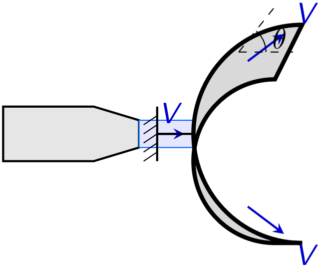
- Frictional loss reduces the efficiency of momentum transfer in practical engineering systems.

Fixed Curved Plate: Deflection Mechanics I

Curved plates, or vanes, are designed to guide the fluid jet along a smooth path, deflecting it through a larger angle without generating high turbulence or stagnation losses. While a flat plate is restricted to a maximum deflection of 90 degrees (reducing the final velocity component in the jet direction to zero), a curved vane can deflect the jet by up to 180 degrees. This permits the fluid to reverse direction, which significantly increases the change in momentum and, consequently, the force exerted on the structure.

- Used in Pelton turbine buckets to reverse flow direction and maximize force.
- Deflection angle is defined relative to the initial jet vector.
- Vanes can have central jet impingement or tangential inlet flow profiles.

Diagram: Jet Striking Center of Symmetrical Fixed Curved Plate I



Fixed Vane

- Symmetric curvature splits the flow equally into top and bottom paths.
- Fluid enters horizontally and leaves the vane tips at angle θ relative to the outlet tangent.
- Nozzle generates jet velocity V; fluid leaves with velocity V (frictionless assumption).

Force Analysis: Symmetrical Fixed Curved Plate I

For a jet striking a symmetrical curved plate at the center, the fluid is deflected smoothly along the profile and exits at the tips at angle θ to the horizontal axis of the jet. Assuming frictionless flow, the exit velocity remains V. The force exerted in the horizontal x-direction (the direction of the jet) is calculated by analyzing the inlet and outlet velocity components: $V_{in} = V$, and $V_{out} =$ -V * $\cos(\theta)$. The negative sign represents the reversal of the horizontal velocity component.

- Change in velocity in x-direction: $\Delta V_x = V_{in} - V_{out} = V - (-V * \cos(\theta)) = V * (1 + \cos(\theta))$.
- Horizontal force exerted by jet: $F_x = \rho * A * V^2 * (1 + \cos(\theta))$.
- Maximum force occurs when $\theta = 0$ (semi-circular vane): $F_x = 2 * \rho * A * V^2$, which is double the flat plate force.
- Due to symmetry, vertical forces cancel out ($F_y = 0$).

Transverse Force and Tangential Flow in Curved Plates I

If the curved vane is asymmetric or the jet strikes tangentially at one of the tips rather than the center, the inlet velocity vector has both horizontal and vertical components. In these configurations, a net transverse force (F_y) is generated in addition to the horizontal force (F_x). For a jet entering tangentially at an angle α to the horizontal and leaving at angle β, the velocity vectors must be resolved along the coordinate axes to find the total momentum change.

- Force in x-direction: $F_x = \rho * A * V * (V * \cos(\alpha) + V * \cos(\beta))$ (for flow reversing direction).
- Force in y-direction: $F_y = \rho * A * V * (V * \sin(\alpha) - V * \sin(\beta))$.
- Tangential entry is utilized in Francis and Kaplan turbine guide vanes to minimize inlet shock losses.

Transverse Force and Tangential Flow in Curved Plates II

- Proper design of inlet and outlet angles is crucial to optimize dynamic efficiency.

Comparative Summary and Engineering Applications I

A comparative analysis of jet impact on fixed flat vs. curved plates demonstrates the clear hydrodynamic advantage of curved profiles. Deflecting the fluid stream through larger angles increases the rate of change of momentum, maximizing the thrust force. While fixed plates do not perform mechanical work because the boundary displacement is zero (Work = Force * Distance = 0), this analysis is the mandatory engineering foundation for designing moving turbine blades and buckets.

- Flat Plate: $F_x = \rho * A * V^2$; Curved Semicircular Vane: $F_x = 2 * \rho * A * V^2$.
- Curved plates are standard in Pelton wheels, where bucket splitters divide the jet and deflect it by nearly 165 to 170 degrees.
- Minimizing shock and separation losses requires aligning the blade tangent to the relative velocity vector.

Course: Fluid Mechanics and Hydraulic Machinery Lecture 25: Analysis of hydrodynamic forces exerted by a fluid jet on stationary vertical flat and curved plates.

- Introduction to impulse-momentum principle
- Force evaluation for stationary plates
- Application in hydraulic turbines (Pelton wheel precursor)

The Impulse-Momentum Principle I

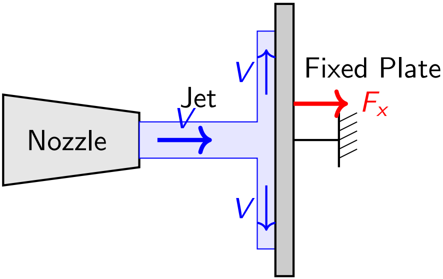
The analysis of hydrodynamic force exerted by a fluid jet on a plate is based on Newton's Second Law of Motion and the Impulse-Momentum Principle. This principle states that the net force acting on a fluid mass is equal to the rate of change of momentum of the fluid in the direction of the force. For a steady flow, the force exerted by the fluid on the plate is equal and opposite to the force exerted by the plate on the fluid, which is calculated as the mass flow rate multiplied by the change in velocity.

- Newton's Second Law: $F = d(m*v)/dt$
- Mass flow rate (m_{dot}) = rho * A * V, where rho is density, A is cross-sectional area of jet, and V is velocity of jet.
- Force exerted by jet in the direction of jet (x-axis): $F_x = m_{dot} * (V_{inletx} - V_{outletx})$.

The Impulse-Momentum Principle II

- Assumptions: No energy loss due to impact, smooth plate surface (no friction), atmospheric pressure throughout.

Diagram: Jet on Fixed Vertical Flat Plate I



- Horizontal jet strikes the flat vertical plate at its center.
- Fluid deflects by 90 degrees, moving parallel to the plate surface.
- The outlet velocity vectors have zero component in the horizontal direction (x-axis).
- A reaction force F_x acts on the plate, balanced by the rigid support.

Force Equation: Fixed Vertical Flat Plate I

Let us derive the force exerted by the jet on a stationary vertical plate. The initial velocity of the jet in the direction of flow (x-axis) is V. Upon striking the plate, the jet is deflected through 90 degrees, meaning its final velocity in the x-direction is zero ($V_{finalx} = 0$). Using the impulse-momentum equation, the force F_x exerted by the jet on the plate in the x-direction is:

F_x = Mass flow rate * (Initial velocity - Final velocity) F_x = (rho * A * V) * (V - 0) F_x = rho * A * V^2

Where: - rho = density of the fluid (e.g., 1000 kg/m^3 for water) - A = cross-sectional area of the jet = pi * d^2 / 4 (d = diameter of jet) - V = velocity of the jet.

- The force F_x is direct, normal to the plate surface.
- Since the plate is stationary, work done by the jet is zero (Work = F_x * u = 0, where u = 0).

Force Equation: Fixed Vertical Flat Plate II

- Kinetic energy of the jet is entirely converted into pressure energy at the stagnation point and then dissipated as turbulence.
- If friction is neglected, the exit velocities of the split jets along the plate remain equal to V.

Fixed Curved Vane: Jet at the Center I

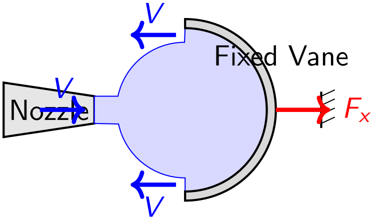
When a jet strikes a fixed curved vane (or plate) at its center, it is deflected along the curved profile and leaves the vane at an angle. The curved plate is symmetrical, meaning the jet splits equally into two streams, each deflecting by an angle theta. The curvature allows for a smoother flow transition, reducing turbulence compared to a flat plate. Because the jet is deflected by more than 90 degrees (often up to 180 degrees), the change in momentum is greater, leading to a significantly higher hydrodynamic force.

- The jet is directed horizontally at the center of the curved plate.
- The vane is smooth; hence, the velocity of fluid along the surface remains V if friction is neglected.
- The deflection angle of the jet is (180 - theta), where theta is the angle made by the outlet tip of the vane with the horizontal.

Fixed Curved Vane: Jet at the Center II

- Symmetrical design ensures that vertical forces cancel each other out ($F_y = 0$).

Diagram: Jet on Fixed Curved Plate I



- The curved profile causes the jet to deflect backward.
- Fluid follows the inner curve smoothly, exiting at angle theta to the horizontal.
- A larger momentum change occurs because the flow direction is reversed.
- The resulting force F_x is higher than that on a flat vertical plate.

Force Equation: Fixed Curved Plate I

Let the jet strike the symmetrical curved plate at the center. The jet is deflected through an angle of (180 - theta), where theta is the angle of the outlet tip of the vane with the horizontal. The initial velocity of the jet in the x-direction is V. Since the plate is smooth, the fluid leaves the outlet tips with the same velocity V. The outlet velocity vector at each tip makes an angle theta with the horizontal, pointing in the negative x-direction. Thus, the outlet velocity component in the x-direction is: $V_{outletx}$ = -V * cos(theta). Using the impulse-momentum equation: F_x = Mass flow rate * (Initial velocity in x - Final velocity in x) F_x = (rho * A * V) * [V - (-V * cos(theta))] F_x = rho * A * V^2 * (1 + cos(theta))

- If the curved plate is semi-circular (theta = 0), then cos(0) = 1, and F_x = 2 * rho * A * V^2.
- The force exerted on a semi-circular curved plate is twice that of a flat vertical plate.

Force Equation: Fixed Curved Plate II

- This force doubling is the core hydrodynamic reason why curved buckets are used in impulse turbines.
- The vertical components of the outlet velocities are V * sin(theta) and -V * sin(theta), which cancel each other out, making net $F_y = 0$.

Tangential Entry on Fixed Curved Plate I

In many actual steam and hydraulic turbines (like the Pelton turbine), the jet does not strike the vane at the center. Instead, it enters tangentially at one of the tips and glides along the inner curved surface, leaving at the other tip. This tangential entry minimizes impact losses (shock losses) at inlet. For a fixed curved plate where the jet enters tangentially at one tip with velocity V, making an angle theta with the horizontal, and leaves at the other tip making an angle phi with the horizontal, we analyze the forces in both x and y directions separately.

- Inlet velocity components: V_{inletx} = V * cos(theta), V_{inlety} = V * sin(theta).
- Outlet velocity components: $V_{outletx}$ = -V * cos(phi), $V_{outlety}$ = V * sin(phi).
- Force in x-direction: F_x = rho * A * V^2 * (cos(theta) + cos(phi)).

Tangential Entry on Fixed Curved Plate II

- Force in y-direction: F_y = rho * A * V^2 * (sin(theta) - sin(phi)).

The comparison between flat and curved stationary plates shows the massive advantage of curved geometries in terms of force generation. A flat vertical plate only destroys the horizontal momentum of the jet, resulting in a force coefficient of 1.0 (relative to rho * A * V^2). A curved plate, by reversing the direction of flow, can achieve a force coefficient up to 2.0 (for a 180-degree turn). In practice, the deflection angle is kept slightly less than 180 degrees (usually around 165 to 170 degrees) to prevent the exiting water jet from interfering with the incoming jet.

- Force ratio: F_{curved} / F_{flat} = 1 + cos(theta). For a semi-circle, this ratio is 2.0.
- Real-world efficiency is reduced slightly by surface friction, which decreases outlet velocity (V_{out} = k * V, where k ¡ 1.0).
- Splashing and turbulence losses are significantly lower in curved plates due to gradual redirection of flow.

Comparative Analysis: Flat vs. Curved Plates II

- These principles are critical for the design of Pelton turbine buckets, where the splitter ridge splits the jet and the curved bowls deflect it.

The study of jet impact on fixed plates is the foundational stepping stone to understanding hydraulic machinery. While stationary plates do not produce mechanical work (since displacement is zero), this analysis forms the basis for analyzing moving plates, which do produce work. These concepts are directly applied in the design and operation of Pelton wheels, impulse steam turbines, and water jet propulsion systems. Key takeaways include the importance of momentum change, the role of boundary geometry, and the optimization of deflection angles to maximize force output without flow interference.

- Hydrodynamic force is directly proportional to fluid density, jet area, and the square of jet velocity.
- Curved surfaces outperform flat surfaces by leveraging flow reversal (up to 200

Engineering Applications & Summary II

- Tangential entry is preferred in turbomachinery to reduce shock losses and guide the flow smoothly.
- Future lectures will extend this analysis to moving plates and calculate hydraulic efficiency.

Course: Fluid Mechanics and Hydraulic Machinery

- Instructor: Professor of Mechanical Engineering
- Lecture 26: Impact of Jets on Moving Plates and Vane Series
- Topic Overview: Impulse Momentum Principle applied to moving systems

Introduction to Moving Vanes I

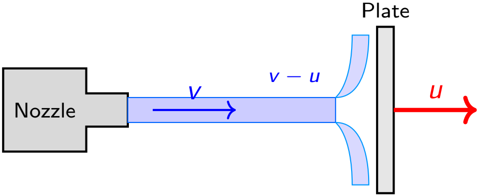
In hydraulic machinery, stationary plates do not yield any mechanical work because their displacement is zero. To extract power from a high-velocity fluid jet, the jet must strike vanes that are free to move in the direction of the force. The impulse-momentum principle dictates that the force exerted by the fluid on the moving plate is proportional to the rate of change of momentum relative to the moving control volume. Understanding single moving vanes provides the mathematical basis for analyzing turbine runners, where kinetic energy is converted into rotational mechanical energy.

- Work done requires both a force and a displacement in the direction of the force.
- The fluid velocity relative to the moving plate (relative velocity) determines the mass flow rate striking the plate.

Introduction to Moving Vanes II

- For a single plate moving at velocity u, the mass of water striking the plate per second is rho*a*(v-u).
- A single moving plate is an idealized concept used to transition to the practical series of vanes.

Jet Striking a Flat Vertical Moving Plate I



- A jet of cross-sectional area 'a' and velocity 'v' strikes a plate moving away at velocity 'u'.
- Relative velocity of the jet with respect to the plate is (v - u).
- Mass of water striking the plate per second is given by $m_{dot} =$ rho * a * (v - u).
- The plate experiences a normal force F_x equal to the rate of change of momentum in the direction of jet motion.

Mathematical Analysis: Single Flat Plate I

To determine the mechanical power generated, we analyze the force and work done on a single moving flat plate. The normal force F_x exerted by the jet is F_x = mass flow rate * change in velocity = rho * a * (v - u) * (v - u) = rho * a * $(v - u)^2$. The work done per second (power) is the product of the force and the plate velocity: W = F_x * u = rho * a * $(v - u)^2$ * u. The input kinetic energy per second is 0.5 * rho * a * v^3. The hydraulic efficiency is the ratio of work done to input energy. Differentiating work done with respect to u shows that the maximum efficiency occurs when $u = v/3$, yielding a maximum theoretical efficiency of only 29.63

- Force on moving plate: F_x = rho * a * $(v - u)^2$ Newtons.
- Work Done per second: W = rho * a * $(v - u)^2$ * u Watts.
- Maximum efficiency occurs at $u = v / 3$.
- Maximum theoretical efficiency of a single flat plate is $8/27 =$ 29.63

Jet Striking a Curved Moving Vane I

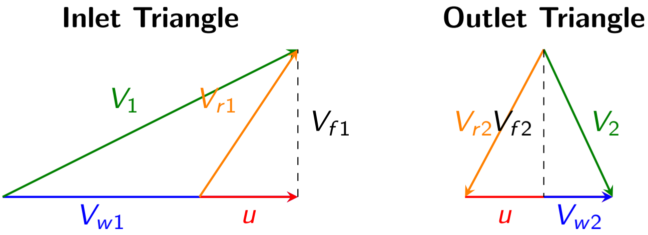
Curved vanes are used instead of flat plates because they deflect the fluid smoothly without shock, minimizing turbulence losses. If the vane is moving with a velocity u in the direction of the jet, the jet enters the vane tangentially with a relative velocity $V_{r1} = V_1$ - u_1 (assuming direction of motion is the same). In the absence of friction, the relative velocity remains constant along the curved profile, so $V_{r2} = V_{r1}$. The absolute velocity of the jet at outlet, V_2, is the vector sum of the relative velocity V_{r2} and the vane velocity u_2. By using curved vanes, we can redirect the fluid backward, which increases the change in momentum and significantly raises the efficiency of the energy transfer.

- Curved profile guides the fluid smoothly, reducing impact/eddy losses.
- Inlet relative velocity: $V_{r1} = V_1$ - u (for horizontal jet and plate motion).

Jet Striking a Curved Moving Vane II

- Outlet relative velocity $V_{r2} = K * V_{r1}$, where K is the blade friction coefficient ($K = 1$ for smooth blade).
- Deflecting the jet through an angle greater than 90 degrees increases momentum transfer.

Inlet and Outlet Velocity Triangles I



- V_1 and V_2 are absolute velocities at inlet and outlet.
- V_{r1} and V_{r2} are relative velocities of the fluid relative to the blade.
- u is the tangential velocity of the vane (assumed $u_1 = u_2 = u$ for axial/tangential flow).
- V_{w1} and V_{w2} are whirl velocities, representing components in the direction of motion.

Work Done and Efficiency for Curved Vane I

The force exerted by the jet in the direction of vane motion is determined by the impulse-momentum principle: F_x = mass flow rate * (change in velocity components in x-direction). The mass flow rate entering the vane is m_{dot} = rho * a * V_{r1} (since the jet strikes the vane with relative velocity V_{r1}). The change in velocity is (V_{w1} - (-V_{w2})) = (V_{w1} + V_{w2}) when the deflection angle is obtuse, or (V_{w1} - V_{w2}) if it is acute. Therefore, F_x = rho * a * V_{r1} * (V_{w1} +/- V_{w2}). The work done per second (power) is W = F_x * u = rho * a * V_{r1} * (V_{w1} +/- V_{w2}) * u. Hydraulic efficiency is this work done divided by the kinetic energy of the incoming jet: eta = [rho * a * V_{r1} * (V_{w1} +/- V_{w2}) * u] / [0.5 * rho * a * V_1^3].

- Force equation: F_x = rho * a * V_{r1} * (V_{w1} +/- V_{w2}) depending on outlet whirl direction.
- Work done per second (Power): W = F_x * u Watts.

Work Done and Efficiency for Curved Vane II

- Hydraulic Efficiency: eta $= 2$ * V_{r1} * (V_{w1} $+$/- V_{w2}) * u / $V_1{}^3$.
- If the vane is smooth and friction is neglected, $V_{r2} = V_{r1}$.

Series of Vanes Mounted on a Wheel I

A single moving plate is not practical for continuous power generation because as the plate moves forward, the jet eventually fails to strike it, and fluid escapes without doing work. In real hydraulic turbines, this is resolved by mounting a series of identical curved vanes along the periphery of a runner wheel. As the wheel rotates, one vane after another comes in front of the jet in rapid succession. Consequently, the jet strikes a vane continuously. Crucially, this means that the entire mass of water leaving the nozzle, $m_{dot} = \text{rho} * a * V_1$, is intercepted by the wheel. The mass flow rate is no longer limited to the relative velocity term rho * a * (V_1 - u) as it was for a single plate.

- A series of vanes ensures continuous energy transfer and no water escapes unutilized.
- Mass flow rate striking the wheel per second is the full nozzle discharge: $m_{dot} = \text{rho} * a * V_1$.

Series of Vanes Mounted on a Wheel II

- This fundamental difference dramatically increases the power and efficiency of the system.
- Forms the working principle of Pelton wheels and other impulse turbines.

Force and Efficiency: Vane Series I

For a series of flat vertical plates mounted on a wheel, the mass flow rate striking the wheel is rho * a * v. The change in velocity of the jet in the direction of motion is (v - u). Therefore, the force exerted on the wheel is F_x = rho * a * v * (v - u). The work done per second is W = F_x * u = rho * a * v * (v - u) * u. The kinetic energy supplied by the jet is 0.5 * rho * a * v^3. The hydraulic efficiency is eta = [rho * a * v * (v - u) * u] / [0.5 * rho * a * v^3] = 2 * u * (v - u) / v^2. To find the condition for maximum efficiency, we differentiate eta with respect to u, which yields u = v/2. Substituting this back, we get the maximum theoretical efficiency of a series of flat plates to be 50

- Force on a series of flat plates: F_x = rho * a * v * (v - u) Newtons.
- Work done per second: W = rho * a * v * (v - u) * u Watts.
- Hydraulic Efficiency: eta = 2 * u * (v - u) / v^2.

Force and Efficiency: Vane Series II

- Maximum efficiency is 50

Comparison and Practical Applications I

Comparing single plates with a series of vanes highlights the transition from theoretical mechanics to practical hydraulic machine design. While a single flat plate has a maximum efficiency of 29.63

- Efficiency increases: Single Flat (29.63
- Pelton turbines deflect the jet by 165-170 degrees to avoid interference with trailing buckets.
- Friction and windage losses reduce practical maximum efficiency to around 85-90
- Understanding these velocity relationships is key to sizing nozzles, runners, and bucket geometry.

Course: Fluid Mechanics and Hydraulic Machinery Lecture 27: Analysis of jet impact on single moving vanes versus a series of vanes mounted on a wheel, focusing on force, work done, and hydraulic efficiency.

- Impact of jet on moving elements
- Single moving plate analysis
- Series of vanes on a rotor
- Kinematics and efficiency optimization

Introduction to Moving Vanes I

In hydraulic turbines, fluid dynamic forces are generated by directing high-velocity jets onto moving surfaces (vanes or buckets). Unlike stationary vanes, moving vanes do work as they displace under the action of the jet force. The analysis requires relative velocity considerations, where the effective velocity striking the vane is the vector difference between the jet velocity and the vane velocity.

- Vanes move at a velocity 'u' in the direction of the jet velocity 'V'.
- Relative velocity V_r = V - u is the velocity with which the fluid strikes the vane.
- The mass flow rate striking a single moving vane is reduced to ρ*A*(V - u).
- Work is done because the point of application of the force moves at velocity 'u'.

Single Moving Flat Vertical Plate I

Consider a jet of area A and velocity V striking a single flat plate moving with velocity u in the direction of the jet. The mass of water striking the plate per second is ρ*A*(V - u). The jet leaves the plate at relative velocity (V - u) parallel to the plate. Thus, the final velocity in the direction of motion is zero. The force exerted is $F = \rho*A*(V - u)^2$. Work done per second is $F * u = \rho*A*(V - u)^2 * u$.

- Mass flow rate striking the plate: $m_{dot} = \rho*A*(V - u)$.
- Force exerted in the direction of motion: $F_x = m_{dot}$ * (Initial Relative Velocity - Final Relative Velocity) $= \rho*A*(V - u)[(V - u) - 0] = \rho*A*(V - u)^2$.
- Work Done per second (Power): $W = F_x * u = \rho*A*(V - u)^2 * u$.
- Note that $W = 0$ when $u = 0$ (stationary plate) or $u = V$ (no impact).

Efficiency of Single Moving Flat Plate I

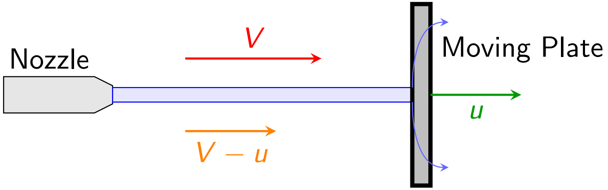
The kinetic energy supplied by the jet per second is 0.5 * ρ * A * V^3. The efficiency (η) is the ratio of work done per second to the kinetic energy supplied. Thus, η = [2 * u * (V - u)^2] / V^3. To find the maximum efficiency, we differentiate η with respect to u and set it to zero, which yields u = V/3. The maximum theoretical efficiency for a single moving flat plate is only 8/27, or approximately 29.63

- Efficiency formula: η = Work Done per second / Input Kinetic Energy = [ρ*A*(V - u)^2 * u] / [0.5 * ρ * A * V^3] = 2*u*(V-u)^2 / V^3.
- Condition for maximum efficiency: $d\eta/du = 0 \Rightarrow$ d/du (u*V^2 - 2*u^2*V + u^3) = 0 $\Rightarrow$ V^2 - 4*u*V + 3*u^2 = 0.
- Solving the quadratic equation gives u = V or u = V/3. Since u = V yields zero efficiency, u = V/3 is the optimum.

Efficiency of Single Moving Flat Plate II

- Maximum efficiency $\eta_{\text{max}} = 2 * (\text{V}/3) * (2*\text{V}/3)^2 \;/\; \text{V}^3 = 8/27 \approx 29.63$

Kinematic Diagram of Single Moving Vane I



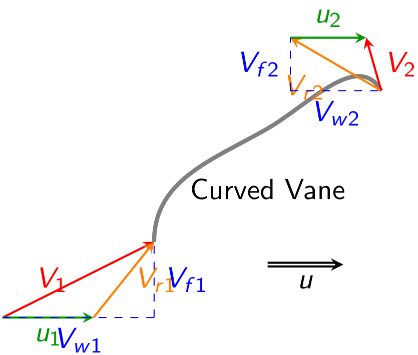
- Nozzle delivers fluid at absolute velocity V.
- Plate moves forward at velocity u.
- Relative velocity striking the plate is V_r = V - u.
- Fluid is deflected along the surface of the plate at relative velocity V_r.

Moving Curved Vane and Velocity Triangles I

When a jet strikes a moving curved vane tangentially, we must construct velocity triangles at the inlet and outlet. The fluid enters the vane with relative velocity V_{r1} = V1 - u1 (vector difference) and leaves with relative velocity V_{r2}. If the vane is smooth, friction is neglected and $V_{r2} = V_{r1}$. The forces are calculated using momentum changes in the tangential (whirl) direction, involving velocities of whirl V_{w1} and V_{w2}.

- Inlet velocity triangle components: V1 (absolute), u1 (vane velocity), V_{r1} (relative), V_{w1} (whirl), V_{f1} (flow).
- Outlet velocity triangle components: V2 (absolute), u2 (vane velocity), V_{r2} (relative), V_{w2} (whirl), V_{f2} (flow).
- Mass flow rate striking a single curved vane: $m_{dot} = \rho$*A*V_{r1}.
- Force exerted in direction of motion: $F_x = \rho$*A*V_{r1} * ($V_{w1} \pm V_{w2}$).

Inlet and Outlet Velocity Triangles I



- V_1 and V_2 are absolute velocities of the jet at inlet and outlet.
- V_{r1} and V_{r2} are relative velocities at inlet and outlet, tangential to the blade.
- V_{w1} and V_{w2} are components of absolute velocity in the direction of vane motion.

Inlet and Outlet Velocity Triangles II

- u_1 and u_2 are the linear velocities of the vane at inlet and outlet.

Series of Vanes Mounted on a Wheel I

In practical hydraulic machines, a single vane is not used because much of the fluid would escape without doing work. Instead, a series of vanes are mounted on the periphery of a rotor or wheel. As the wheel rotates, successive vanes come in front of the jet. Consequently, the entire mass of water leaving the nozzle, $m_{dot} = \rho$*A*V, strikes the wheel. This fundamental difference drastically improves the mass utilization and power output of the system.

- Mass flow rate striking the wheel per second: $m_{dot} = \rho$*A*V (where V is jet velocity).
- No fluid is wasted; the system operates in a continuous steady-state.
- Force exerted in direction of motion: $F_x = \rho$*A*V * $(V_{w1} \pm V_{w2})$.
- This forms the thermodynamic and kinematic foundation for impulse turbines (e.g., Pelton Wheel).

Efficiency of Series of Flat Vanes I

For a series of flat plates mounted radially on a wheel and struck normally by a jet: the force is $F_x = \rho$*A*V*(V - u). Work done per second is F_x * u = ρ*A*V*(V - u)*u. Kinetic energy supplied is 0.5*ρ*A*V^3. The efficiency is η = [2*u*(V - u)]/V^2. Setting the derivative $d\eta/du = 0$ gives u = V/2. Substituting this back yields a maximum efficiency of 50

- Force: $F_x = \rho$*A*V*(V - u), since mass flow rate is ρ*A*V and change in velocity is (V - u).
- Work Done per second: W = ρ*A*V*(V - u)*u.
- Efficiency: η = [2*u*(V - u)] / V^2.
- Maximum efficiency occurs at u = V/2 and is equal to η_{max} = 2*(V/2)*(V/2) / V^2 = 50

Comparison: Single Vane vs. Series of Vanes I

This comparison highlights the transition from a single moving obstruction to a continuous machine. In a single vane, mass flow rate is ρ*A*(V-u) because the vane moves away from the source, leading to transient operation and low efficiency (max 29.63

- Mass flow rate: Single Vane is ρ*A*(V-u), whereas Series is ρ*A*V.
- Maximum efficiency (flat plate): Single Vane is 29.63
- Maximum efficiency (curved plate/vane): Single Vane is 59.26
- Engineering applications: Series of vanes represent the basic design of turbines, whereas single vanes are only used in simple theoretical dynamics.

Impact of jet on curved vanes with special reference to turbines & pumps I

Course: Fluid Mechanics and Hydraulic Machinery (ME-302)
Lecture 28: Impact of Jet on Curved Vanes with Special Reference to Turbines & Pumps
This lecture covers the fundamental hydrodynamic principles of jet interaction with curved vanes, the construction of velocity triangles at inlet and outlet, work done, efficiency, and their direct application in the design of impulse turbines (Pelton wheels) and centrifugal pumps.

- Core Hydrodynamic Principles
- Velocity Triangles Analysis
- Turbomachinery Design Applications

Physics of Jet Impact on Curved Vanes I

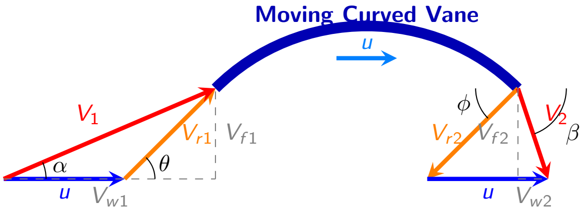
When a high-velocity fluid jet strikes a curved vane, it undergoes a change in direction, which generates an impulse force on the vane according to Newton's Second Law of Motion and the Impulse-Momentum Principle. Unlike flat plates, curved vanes allow for smooth fluid flow redirection without sudden shock, minimizing turbulence and kinetic energy losses. If the vane is moving, the fluid enters and leaves with a relative velocity that is tangent to the vane profile at the inlet and outlet, ensuring shockless entry. The force exerted by the jet in the direction of vane motion is determined by the change in the whirl component of absolute velocity (V_w).

- Impulse-Momentum Principle: Force is proportional to the rate of change of momentum.
- Shockless Entry: The relative velocity vector at inlet must be tangent to the vane tip.

Physics of Jet Impact on Curved Vanes II

- Whirl Velocity (V_w): The tangential component of absolute velocity responsible for torque and work.
- Flow Velocity (V_f): The radial or axial component responsible for fluid throughput.

Velocity Triangles for a Moving Curved Vane I



- V_1, V_2: Absolute velocities at inlet and outlet.
- V_{r1}, V_{r2}: Relative velocities of the fluid relative to the vane.
- u: Linear velocity of the vane (assumed constant for flat translation).
- V_{w1}, V_{w2}: Whirl components at inlet and outlet.
- alpha, beta: Guide vane angle and absolute discharge angle.
- theta, phi: Vane angles at inlet and outlet.

Inlet and Outlet Velocity Triangles Analysis I

Velocity triangles are vector diagrams that represent the relationship between absolute velocity (V), relative velocity (V_r), and peripheral blade speed (u) at the inlet and outlet of the vane. At the inlet, the jet strikes the vane with absolute velocity V_1 at an angle alpha. The relative velocity V_{r1} is the vector difference (V_1 - u_1) and enters the vane at angle theta. If the vane is smooth, the relative velocity remains nearly constant in magnitude ($V_{r2} = k * V_{r1}$, where k is the friction coefficient, k = 1 for frictionless vanes). At the outlet, the relative velocity V_{r2} leaves the vane tangent to the outlet profile at angle phi. The absolute outlet velocity V_2 is the vector sum ($V_{r2} + u_2$) and leaves at angle beta.

- Friction Coefficient (k): Represents losses along the vane surface ($V_{r2} = k * V_{r1}$).
- Whirl Velocity Change: Delta $V_w = V_{w1} + V_{w2}$ (if jet is deflected by ¿ 90 deg) or V_{w1} - V_{w2} (if ¡ 90 deg).

Inlet and Outlet Velocity Triangles Analysis II

- Shockless Flow: The design must align the blade angles (theta and phi) with relative velocity vectors.
- Blade Speeds: For a single translator vane, $u_1 = u_2 = u$; for radial flow (turbines/pumps), $u_1 \mathrel{!=} u_2$.

Work Done and Hydraulic Efficiency I

The rate of work done by the jet on a series of moving curved vanes is derived from Euler's turbomachinery equation. The force exerted in the tangential direction is F_x = rho * a * V_1 * (V_{w1} +- V_{w2}), where rho is fluid density and a is jet area. The sign depends on whether the absolute velocity at outlet is deflected backwards (+ sign, beta ¡ 90 deg) or forwards (- sign, beta ¿ 90 deg). The power developed (Work Done per second) is Power = F_x * u = rho * a * V_1 * (V_{w1} +- V_{w2}) * u. The kinetic energy supplied by the jet per second is 0.5 * rho * a * V_1^3. The hydraulic efficiency (eta_h) is the ratio of work done to the kinetic energy supplied.

- Force Equation: F_x = m_{dot} * (V_{w1} - V_{w2}) in vector form, where m_{dot} = rho * a * V_1.
- Work Done per Unit Weight: W = (1/g) * (V_{w1} * u_1 +- V_{w2} * u_2), which is the general Euler Turbine Equation.

Work Done and Hydraulic Efficiency II

- Maximum Efficiency: For a semi-circular vane (180 deg deflection) and zero friction, max efficiency is 100
- Energy Losses: Mainly due to friction along the vane surface and residual kinetic energy of the leaving jet (V_2^2 / 2).

Radial Flow Vanes and Turbomachinery Applications I

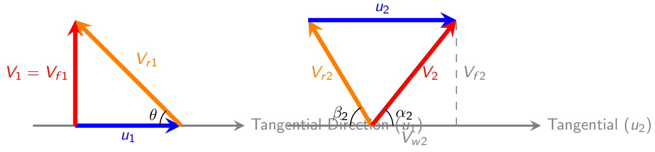
In actual hydraulic machines like Pelton turbines, Francis turbines, and centrifugal pumps, the fluid flow is rarely purely rectilinear. In radial-flow machines, the fluid enters at a different radius (R_1) than it exits (R_2). Because the peripheral speed of the impeller varies with radius (u = omega * R), the blade speeds at inlet and outlet are different (u_1 != u_2). For turbines, fluid enters at the outer diameter and exits at the inner diameter (inward radial flow), converting pressure and kinetic energy into mechanical shaft work. For centrifugal pumps, the fluid enters at the eye (inner radius) and is thrown outward to the outer radius (outward radial flow), increasing the kinetic and pressure energy of the fluid.

- Pelton Wheel: An impulse turbine utilizing double-hemispherical buckets (curved vanes) with deflection angles of 165-170 degrees.

Radial Flow Vanes and Turbomachinery Applications II

- Francis Turbine: A reaction turbine with radial inlet and axial outlet, using guided curved runner blades.
- Centrifugal Pump Impeller: Comprises backward-curved vanes to achieve stable head-discharge characteristics and high efficiency.
- Euler Head: $H_e = (V_{w1} * u_1 - V_{w2} * u_2) / g$ (for turbines) and $(V_{w2} * u_2 - V_{w1} * u_1) / g$ (for pumps).

Radial Flow Impeller Velocity Triangles I



Inlet Triangle ($V_{w1} = 0$) **Outlet Triangle** (Backward-Curved)

- Radial Inlet: Entering absolute velocity V_1 has no whirl component (alpha = 90 deg, V_{w1} = 0).
- Backward-Curved Outlet: Vane outlet angle $beta_2$ is acute, providing stable operating conditions.
- u_1, u_2: Peripheral speeds at inlet and outlet radii (u_1 ¡ u_2 for pumps).
- V_{w2}: Whirl velocity component at outlet that directly determines fluid head generation.

Jet Impact in Centrifugal Pump Impellers I

In centrifugal pumps, the impeller vanes are designed to guide the fluid outwards while imparting mechanical energy to it. Since the pump is a work-absorbing machine, the energy transfer is the reverse of a turbine. Fluid enters the impeller eye radially (meaning absolute velocity V_1 has no whirl component, $V_{w1} = 0$, $alpha_1 =$ 90 deg). The impeller rotates, and the vanes do work on the liquid. To minimize turbulence, friction, and separation losses, backward-curved vanes (outlet blade angle $beta_2$ ¡ 90 deg) are universally preferred. This design yields a rising head-discharge curve, which provides stable control and prevents overloading of the motor at high flow rates.

- Backward-Curved Vanes: Vane outlet angle $beta_2$ is less than 90 degrees, typically 20 to 30 degrees.
- Energy Transfer: Work done per second on the fluid is rho * Q * V_{w2} * u_2, since $V_{w1} = 0$.

Jet Impact in Centrifugal Pump Impellers II

- Manometric Efficiency: Ratio of the manometric head (actual lift + pipe losses) to the Euler head.
- Vane Exit Conditions: High relative velocity at outlet is converted to static pressure in the volute casing or diffuser.

Shockless Entry and Blade Design Parameters I

For maximum efficiency and durability, turbomachinery blades must be designed for shockless entry of fluid. This condition requires that the relative velocity vector of the fluid at the inlet (V_{r1}) must align perfectly with the tangent to the vane profile at the inlet tip. Any mismatch results in shock losses, flow separation, eddies, and accelerated blade erosion due to cavitation. In design, the inlet blade angle theta is calculated as theta = arctan(V_{f1} / (V_{w1} - u_1)). Similarly, at the outlet, the vane angle phi is designed based on the desired velocity triangles to ensure a smooth discharge of fluid with minimum residual kinetic energy.

- Vane Inlet Angle (theta): Designed such that tan(theta) = V_{f1} / (V_{w1} - u_1) for shockless entry.
- Vane Outlet Angle (phi): Designed to control the whirl velocity V_{w2} and the absolute outlet velocity V_2.

Shockless Entry and Blade Design Parameters II

- Cavitation Risk: Occurs when pressure drops below vapor pressure, often triggered by flow separation from poor blade alignment.
- Aerofoil Profiles: Modern hydraulic machines use aerofoil blade designs instead of simple curved plates for high lift-to-drag ratios.

Summary of Engineering Equations I

This lecture summarized the critical mathematical formulations for jet impact on curved vanes and their application to hydraulic machines. The primary performance metrics are work done, power output, and hydraulic efficiency. Understanding the velocity triangles is fundamental to the design of both power-producing machines (Pelton, Francis, and Kaplan turbines) and work-absorbing machines (centrifugal and axial pumps). By manipulating the blade angles (theta and phi), engineers can optimize machines for specific head, discharge, and rotational speed requirements, conforming to the laws of thermodynamic similarity.

- Euler's Equation: Work = m_{dot} * (V_{w1} * u_1 +- V_{w2} * u_2) is the cornerstone of turbomachinery design.
- Pelton Bucket Deflection: Maximum work is obtained when the deflection angle is close to 180 degrees.

Summary of Engineering Equations II

- Pump Stability: Backward-curved vanes provide stable operation curves, whereas forward-curved vanes are unstable.
- Efficiency Optimization: Minimizing exit kinetic energy (V_2^2 / 2g) is key to maximizing hydraulic efficiency.

Impact of Jet on Curved Vanes: Turbines & Pumps I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 29: Impact of jet on curved vanes with special reference to turbines & pumps

An advanced examination of hydrodynamic forces, velocity triangles, momentum transfer, and efficiency optimization in curved vanes for impulse turbines and centrifugal pump impellers.

- Hydrodynamic force derivation on curved vanes
- Inlet and outlet velocity triangles
- Euler's Turbomachine Equation
- Applications to Pelton, Francis turbines, and Centrifugal Pumps

Physics of Hydrodynamic Forces on Curved Vanes I

When a high-velocity fluid jet strikes a curved vane, it is deflected smoothly along the curved path. According to the impulse-momentum principle, the force exerted by the fluid on the vane is equal to the rate of change of momentum of the fluid in the direction of force. Curved vanes are preferred over flat plates in turbomachinery because they prevent shock losses at entry and allow for a larger change in the velocity vector, yielding significantly higher work transfer efficiency.

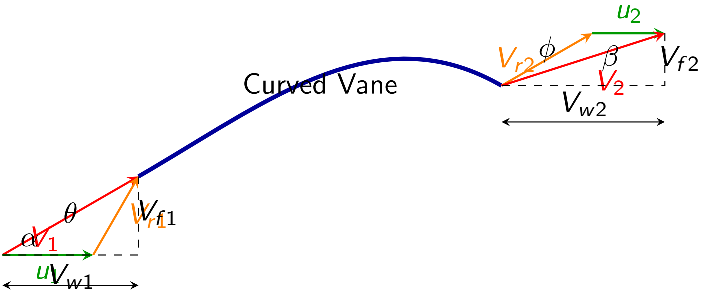
- The momentum equation is derived from Newton's Second Law applied to a control volume enclosing the vane.
- Smooth curvature guides the jet flow, transforming kinetic energy into mechanical torque with minimal flow separation.
- Friction along the vane surface reduces the relative velocity: $V_{r2} = k * V_{r1}$, where k is the vane friction coefficient (k ¡ 1).

Jet Striking Symmetrically at the Center of a Stationary Vane I

For a stationary curved vane where the jet strikes at the center, the jet splits symmetrically into two streams. Each stream is deflected through an angle theta at the vane outlet tips. In the ideal, frictionless case ($k = 1$), the magnitude of velocity remains constant ($V_1 = V_2 = V$). The net hydrodynamic force is computed by integrating the change in momentum along the coordinate axes.

- Mass flow rate striking the vane: $m_{dot} = \rho * A * V$, where ρ is fluid density and A is the jet area.
- Inlet velocity vector: $V_{in} = V * i$. Outlet velocity vector: $V_{out} = -V * \cos(\theta) * i \pm V * \sin(\theta) * j$.
- Force in direction of jet: $F_x = \rho * A * V^2 * (1 + \cos(\theta))$.
- Force perpendicular to jet: $F_y = 0$ due to symmetrical splitting of the jet.

Velocity Triangles for a Moving Curved Vane I



- Inlet velocity triangle links absolute velocity (V_1), relative velocity (V_{r1}), and vane velocity (u_1).
- Outlet velocity triangle links absolute velocity (V_2), relative velocity (V_{r2}), and vane velocity (u_2).
- The whirl components (V_{w1}, V_{w2}) lie along the direction of motion and do the useful work.

Euler Turbomachine Equation & Work Done I

For a series of curved vanes mounted on a runner, the fluid enters and leaves at different radii. Applying the angular momentum equation yields the Euler Turbomachine Equation. The torque exerted on the runner is $T = \rho * Q * (V_{w1} * R_1 \pm V_{w2} * R_2)$, where Q is the volumetric flow rate. The work done per second (Power) is $W = T * \omega = \rho * Q * (V_{w1} * u_1 \pm V_{w2} * u_2)$, which is the basis for all rotary hydraulic machines.

- Euler's equation is independent of the thermodynamic state of the fluid or the internal losses.
- The positive sign (+) is used when the outlet angle β is acute (direction of V_{w2} is opposite to u_2).
- The negative sign (-) is used when the outlet angle β is obtuse (direction of V_{w2} is in the same direction as u_2).

Hydraulic Efficiency of Vanes in Turbines I

For a tangential flow runner (like Pelton wheel) where inlet and outlet velocities are at the same radius, $u_1 = u_2 = u$. The hydraulic efficiency is η_h = Work Done / Kinetic Energy of Jet = [2 * u * ($V_{w1} \pm V_{w2}$)] / $V_1{}^2$. Under ideal conditions where relative velocity magnitude is conserved ($V_{r2} = V_{r1}$) and the inlet jet is tangential ($\alpha = 0$, $V_{w1} = V_1$), maximum efficiency is achieved at a specific speed ratio.

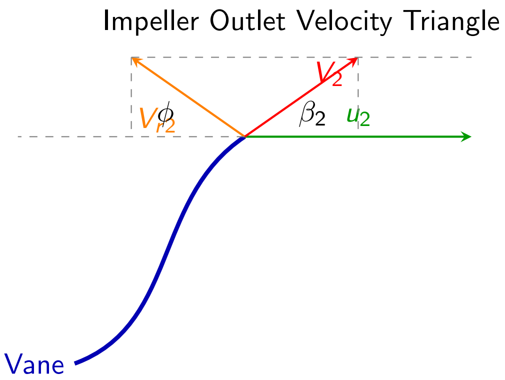
- Optimal speed ratio: $u = V_1$ / 2. At this ratio, the blade speed is half the jet velocity.
- Maximum theoretical efficiency: $\eta_{max} = (1 + \cos(\phi))$ / 2, where ϕ is the vane exit angle.
- For semi-circular buckets ($\phi = 180$ degrees), theoretical efficiency is 100

Curved Vanes in Centrifugal Pumps: Vane Curvature Types I

In centrifugal pumps, work is transferred from the impeller vanes to the fluid to increase its pressure and kinetic energy. Pump impellers are classified based on the vane angle at outlet (ϕ or β_2) into backward-curved, radial, and forward-curved vanes.
Backward-curved vanes are universally preferred in commercial pumps because they offer self-regulating power characteristics and stable head-discharge behavior.

- Backward-curved vanes (ϕ ¡ 90 degrees) yield lower outlet kinetic energy, reducing conversion losses in the volute casing.
- Forward-curved vanes (ϕ ¿ 90 degrees) provide a high head but suffer from steep drop in efficiency and unstable operation.
- Euler's head equation for pumps: $\mathrm{H_e} = (\mathrm{V_{w2}} * \mathrm{u_2}) / \mathrm{g}$ (assuming radial inlet flow where $\mathrm{V_{w1}} = 0$).

Impeller Outlet Velocity Triangle for Backward-Curved Vanes I



- The diagram shows the vector summation: $V_2 = u_2 + V_{r2}$.
- For backward-curved vanes, V_{w2} is less than u_2, reducing the kinetic head to be converted to pressure.

Impeller Outlet Velocity Triangle for Backward-Curved Vanes II

- The blade angle phi represents the backward angle of the impeller blade at the outer periphery.

Vane Design Criteria in Turbines: Pelton vs. Francis I

Curved vanes must be carefully shaped to minimize losses. In Pelton (impulse) turbines, the buckets have a splitter ridge in the center and are shaped like double-hemispherical cups to smoothly guide the jet sideways. In Francis (reaction) turbines, the blades are three-dimensionally curved (mixed-flow) to handle the pressure drop along the flow path while preventing local cavitation near the low-pressure outlet.

- Pelton splitters prevent unbalanced radial forces and ensure symmetrical flow deflection.
- Francis guide vanes regulate the flow angle α to align with the runner blade inlet angle θ to eliminate shock losses.
- Modern CFD is used to optimize the thickness profile and curvature of blades to prevent flow separation.

Design Optimization & Cavitation Limits I

Designing high-performance vanes requires balancing energy extraction against mechanical stress and cavitation. Cavitation occurs when local pressure drops below the fluid's vapor pressure, causing vapor bubbles to form and violently collapse, eroding the vane surface. Mechanical engineers use dimensionless parameters like Specific Speed and Net Positive Suction Head (NPSH) to define the operating limits of these curved vanes.

- Cavitation primarily affects the low-pressure backside of pump vanes and reaction turbine exit zones.
- Correct blade curvature reduces localized velocity peaks, maintaining the pressure above the vapor pressure limit.
- Frictional loss (h_f) is minimized by reducing surface roughness and maintaining boundary layer attachment.

Simple Numerical problems on work done and efficiency. I

Course: Fluid Mechanics and Hydraulic Machinery Lecture 30: Analysis of Work Done, Power Developed, and Efficiencies of Hydraulic Turbines through Solved Examples

- Introduction to energy transfer in hydraulic machines.
- Definitions of hydraulic, mechanical, and overall efficiencies.
- Applying Euler's turbine equation to solve practical problems.

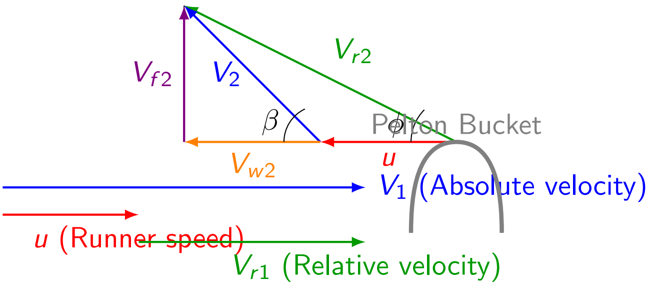
Fundamental Equations of Energy Transfer I

The energy transfer in hydraulic machinery is governed by Euler's turbine equation, which relates the change in angular momentum of the fluid to the work done on or by the rotor. For a runner, the work done per second (power) is given by $W = \rho Q(V_{w1}u_1 \pm V_{w2}u_2)$, where ρ is fluid density, Q is discharge, V_w represents whirl velocity components, and u represents runner tangential velocities at inlet (1) and outlet (2). The positive sign applies to inward flow machines, while the negative sign is standard. Efficiency definitions compare this work done to the net head available at the turbine inlet (H) or the shaft output power.

- Euler's turbine equation is derived from the impulse-momentum principle applied to a rotating control volume.
- Whirl velocity (V_w) is the component of absolute velocity (V) in the tangential direction of runner rotation.

Fundamental Equations of Energy Transfer II

- Work done per unit weight of fluid (Eulerian head) is given by $H_e = \frac{1}{g}(V_{w1}u_1 \pm V_{w2}u_2)$.
- Hydraulic efficiency (η_h) is the ratio of Eulerian head to the net head (H) available at the turbine inlet: $\eta_h = H_e/H$.



- At the inlet, the jet is tangent to the runner, making the absolute velocity V_1, whirl velocity V_{w1}, and relative velocity V_{r1} collinear.
- Relative velocity at inlet is given by $V_{r1} = V_1 - u$, where u is the blade speed.
- At the outlet, the relative velocity V_{r2} leaves at the bucket outlet angle ϕ.

Velocity Triangles for a Pelton Wheel Bucket II

- If friction is neglected, $V_{r2} = V_{r1}$. Otherwise, $V_{r2} = kV_{r1}$ where $k < 1$ represents the bucket friction coefficient.

Numerical Problem 1: Pelton Turbine Work Done I

Problem Statement: A Pelton wheel is supplied with water under a net head of 300 m. The bucket speed is 15 m/s and the discharge is 0.15 m^3/s. The buckets deflect the jet through an angle of 165°. If the coefficient of velocity (C_v) for the nozzle is 0.98, determine the power developed by the runner and the hydraulic efficiency of the turbine. Assume bucket friction coefficient $k = 1$.

- First, calculate the theoretical jet velocity: $V_1 = C_v\sqrt{2gH} = 0.98 \times \sqrt{2 \times 9.81 \times 300} = 75.19$ m/s.
- Determine relative velocity at inlet: $V_{r1} = V_1 - u = 75.19 - 15 = 60.19$ m/s.
- Calculate bucket deflection angle parameters: deflection angle $\theta = 165°$, hence bucket outlet angle $\phi = 180° - 165° = 15°$.
- Compute relative velocity at outlet: $V_{r2} = V_{r1} = 60.19$ m/s (since $k = 1$).

Numerical Problem 1: Pelton Turbine Work Done II

- Find whirl velocity at outlet:
  $V_{w2} = V_{r2} \cos\phi - u = 60.19 \cos(15^\circ) - 15 = 43.14$ m/s.
  Since V_{w2} is positive, the outlet relative velocity horizontal component exceeds u, making the absolute velocity exit backwards.

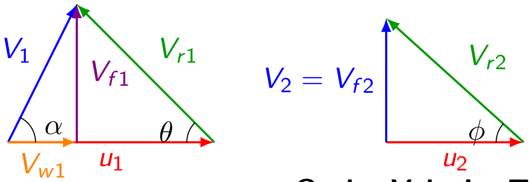
Solution Analysis of Pelton Turbine Problem I

Continuing the calculation: The work done per second (power) is $P = \rho Q(V_{w1} + V_{w2})u$ because the jet is deflected through an angle greater than 90° and the absolute outlet velocity is directed backwards, meaning V_{w2} acts opposite to the direction of blade speed u in a vector sense. Substituting the values: $P = 1000 \times 0.15 \times (75.19 + 43.14) \times 15 = 266.24$ kW. Next, we calculate the hydraulic efficiency by comparing this runner power to the total water power available at the inlet.

- Water Power available at inlet: $P_{\text{water}} = \rho g Q H = 1000 \times 9.81 \times 0.15 \times 300 = 441.45$ kW.
- Runner Power developed: $P_{\text{runner}} = \rho Q(V_{w1} + V_{w2})u = 1000 \times 0.15 \times (75.19 + 43.14) \times 15 = 266.24$ kW.
- Hydraulic Efficiency: $\eta_h = P_{\text{runner}}/P_{\text{water}} = 266.24/441.45 = 60.31\%$.

Solution Analysis of Pelton Turbine Problem II

- Discussion: The efficiency is lower than typical Pelton designs ($> 85\%$) because the speed ratio $u/V_1 = 15/75.19 \approx 0.20$ is far from the optimum speed ratio of 0.46 to 0.48.



Inlet Velocity Triangle

Outlet Velocity Triangle (Radial Discharge, $V_{w2} = 0$)

- In reaction turbines (Francis/Kaplan), water enters the runner with both pressure and kinetic energy.
- The inlet velocity triangle contains the whirl component V_{w1} and the flow component V_{f1} at guide vane angle α.
- The outlet velocity triangle is usually designed for radial discharge, which means $\beta = 90^\circ$ and $V_{w2} = 0$.
- Radial discharge at outlet minimizes the kinetic energy loss at the draft tube inlet, maximizing hydraulic efficiency.

Numerical Problem 2: Francis Turbine Efficiency I

Problem Statement: An inward flow reaction turbine (Francis turbine) operates under a net head of 12 m. The guide vane angle $\alpha = 20^\circ$ and the runner blade angle at inlet $\theta = 60^\circ$. The radial velocity of flow is constant and equal to 4 m/s. The outlet discharge is radial (whirl velocity at outlet $V_{w2} = 0$). Find: (1) The tangential velocity of the runner at inlet (u_1), (2) The work done per unit weight of water, and (3) The hydraulic efficiency (η_h) of the turbine.

- Identify known values: Net Head $H = 12$ m, $\alpha = 20^\circ$, $\theta = 60^\circ$, $V_{f1} = V_{f2} = 4$ m/s.
- Calculate the whirl velocity at inlet: $V_{w1} = V_{f1}/\tan\alpha = 4/\tan(20^\circ) = 10.99$ m/s.
- Use the inlet velocity triangle relation: $\tan\theta = V_{f1}/(V_{w1} - u_1)$ to isolate the blade speed u_1.

Numerical Problem 2: Francis Turbine Efficiency II

- Solve for u_1:
  $u_1 = V_{w1} - (V_{f1}/\tan\theta) = 10.99 - (4/\tan 60^\circ) = 8.68$ m/s.

Continuing the calculation: With radial discharge, the outlet velocity triangle has $V_{w2} = 0$. The Euler turbine equation simplifies to the work done per unit weight (Eulerian Head H_e): $H_e = \frac{V_{w1} u_1}{g}$. Substituting the calculated values gives:
$H_e = \frac{10.99 \times 8.68}{9.81} = 9.725$ m. The hydraulic efficiency is the ratio of this Eulerian Head to the net head: $\eta_h = \frac{H_e}{H} = \frac{9.725}{12} = 81.04\%$. This efficiency indicates a well-designed runner operating close to its design point under the given flow conditions.

- Radial discharge at outlet means $V_2 = V_{f2}$ and $V_{w2} = 0$, which is a standard design condition to maximize energy extraction.
- Eulerian Head ($H_e = 9.725$ m) represents the specific energy transferred from fluid to runner.

Solution Analysis of Francis Turbine Problem II

- Hydraulic efficiency of 81.04% shows that 18.96% of the inlet head is lost to friction, leakage, and residual kinetic energy at the exit ($V_2^2/2g$).
- Residual kinetic energy loss: $V_2^2/2g = V_{f2}^2/2g = 4^2/(2 \times 9.81) = 0.815$ m, which represents 6.8% of the total available head.

Common Mistakes in Work & Efficiency Problems I

Solving numerical problems in hydraulic machinery requires careful attention to signs, velocity components, and efficiency definitions. A common mistake is using the absolute velocity (V) instead of the whirl velocity (V_w) in the Euler equation. Additionally, students often confuse the blade angle (tangent to the blade) with the absolute flow angle (angle of the absolute velocity vector). Finally, neglecting the sign of V_{w2} when the outlet triangle changes from acute to obtuse blade angles can lead to incorrect power calculations.

- Always draw velocity triangles at both inlet and outlet before starting calculations.
- Verify the sign of V_{w2}: if the outlet velocity triangle has an obtuse angle $\beta > 90^\circ$, V_{w2} is negative, making the work formula $\rho Q(V_{w1} + V_{w2})u$.

Common Mistakes in Work & Efficiency Problems II

- Distinguish between Net Head (H) and Gross Head (H_g): $H = H_g - h_f$, where h_f is pipeline friction loss.
- Mechanical efficiency (η_m) accounts for bearing friction and windage losses: $\eta_m = P_{\text{shaft}}/P_{\text{runner}}$.

Summary of Key Performance Formulas I

To summarize Lecture 30, the performance of hydraulic turbines can be predicted using a concise set of mathematical relations derived from the velocity triangles. The overall efficiency (η_o) is the product of hydraulic efficiency (η_h), mechanical efficiency (η_m), and volumetric efficiency (η_v), i.e., $\eta_o = \eta_h \times \eta_m \times \eta_v$. This efficiency chain links the raw fluid energy at the inlet to the final mechanical power output at the turbine shaft.

- Eulerian Head: $H_e = \frac{1}{g}(V_{w1}u_1 \pm V_{w2}u_2)$. For radial discharge, $H_e = \frac{V_{w1}u_1}{g}$.
- Water Power: $P_w = \rho g Q H$. Runner Power: $P_r = \rho Q(V_{w1}u_1 \pm V_{w2}u_2)$. Shaft Power: $P_s = \eta_m P_r$.
- Hydraulic Efficiency: $\eta_h = H_e/H$. Overall Efficiency: $\eta_o = P_s/P_w$.
- Speed ratio ($K_u = u_1/\sqrt{2gH}$) and Flow ratio ($K_f = V_{f1}/\sqrt{2gH}$) are key non-dimensional design parameters.